EDF R&D

Fluid Dynamics, Power Generation and Environment Department
Single Phase Thermal-Hydraulics Group

6, quai Watier
F-78401 Chatou Cedex

Tel: 33 1 30 87 75 40
Fax: 33 1 30 87 79 16

MAY 2020

Code_Saturne documentation

Code_Saturne version 6.0 tutorial: three 2D disks

contact: saturne-support@edf.fr

TABLE OF CONTENTS

I Introduction 3

1 Introduction . . . 4
1.1 *Code_Saturne* short presentation . . . 4
1.2 About this document . . . 4
1.3 *Code_Saturne* copyright informations . . . 4

II Three 2D disks 5

1 Study description . . . 6
1.1 Objective . . . 6
1.2 Study creation and preparation . . . 6
1.3 Geometry . . . 7
1.4 Meshes characteristics . . . 7
1.5 Data settings for solid and fluid domains . . . 9
2 Computations of the 3 2D disks configuration . . . 10
2.1 Time stepping parameters . . . 10
2.2 Output management . . . 10
2.3 Results for the uncoupled cases . . . 10
2.4 Parameters for the coupled computation . . . 11
2.5 Results for the coupled computation . . . 12

III Step by step description 14

1 Detailed tutorial step by step . . . 15
1.1 Creation of the study in a terminal . . . 15
1.2 Preparing and launching SYRTHES computation alone . . . 16
1.3 Preparing and launching the *Code_Saturne* computation alone . . . 24
1.4 Preparing and launching *Code_Saturne*-SYRTHES coupled computation . . . 32

Part I

Introduction

1 Introduction

1.1 *Code_Saturne* short presentation

Code_Saturne is a system designed to solve the Navier-Stokes equations in the cases of 2D, 2D axisymmetric or 3D flows. Its main module is designed for the simulation of flows which may be steady or unsteady, laminar or turbulent, incompressible or potentially dilatable, isothermal or not. Scalars and turbulent fluctuations of scalars can be taken into account. The code includes specific modules, referred to as "specific physics", for the treatment of lagrangian particle tracking, semi-transparent radiative transfer, gas, pulverized coal and heavy fuel oil combustion, electricity effects (Joule effect and electric arcs) and compressible flows. *Code_Saturne* relies on a finite volume discretization and allows the use of various mesh types which may be hybrid (containing several kinds of elements) and may have structural non-conformities (hanging nodes).

1.2 About this document

The present document is a tutorial for *Code_Saturne* version 6.0. It presents test cases and guides the future *Code_Saturne* user step by step into the preparation and the computation of the cases. It focuses on *Code_Saturne* SYRTHES coupling feature allowing to run simulations taking into account conjugate heat transfers.

The test case directories, containing the necessary meshes and data are available in the `examples/4-2Ddisks` directory in *Code_Saturne* source directory.

This tutorial focuses on the procedure and the preparation of the *Code_Saturne* and SYRTHES computations with or without SALOME. For more elements on the structure of the code and the definition of the different variables, it is higly recommended to refer to the user manual.

The first part is this introduction. In the second part, the configuration, geometry, data settings, numerical parameters are described for a solid computation alone, a fluid computation alone and finally for a coupled fluid/solid computation. In the third part, a detailed description of the steps to be followed is given.

1.3 *Code_Saturne* copyright informations

Part II

Three 2D disks

1 Study description

1.1 Objective

This tutorial case focuses on *Code_Saturne* coupling feature with SYRTHES, an open source code solving thermal conduction and radiative transfer problems in solids, which is one of the way to model conjugate transfer phenomenon with *Code_Saturne*.

The aim is to simulate the natural convection flow of air inside a sheath containing three electric wires. A 2D cross section of the sheath is considered. Thermal conduction inside the three wires is solved with SYRTHES and coupled with the flow solved with *Code_Saturne*.

In a first step, uncoupled solid and fluid calculations will be set and run separately, and only in a second step, a coupled solid/fluid calculation. The data settings used for the uncoupled step can largely serve as basis for the coupled step.

1.2 Study creation and preparation

First, create the 🗀`3_2D_DISKS` study directory, with two subdirectories 🗀`FLUID` and 🗀`SOLID`. Two ways are available:

- Create coupled study with SALOME module CFDSTUDY as described on Figure II.1.

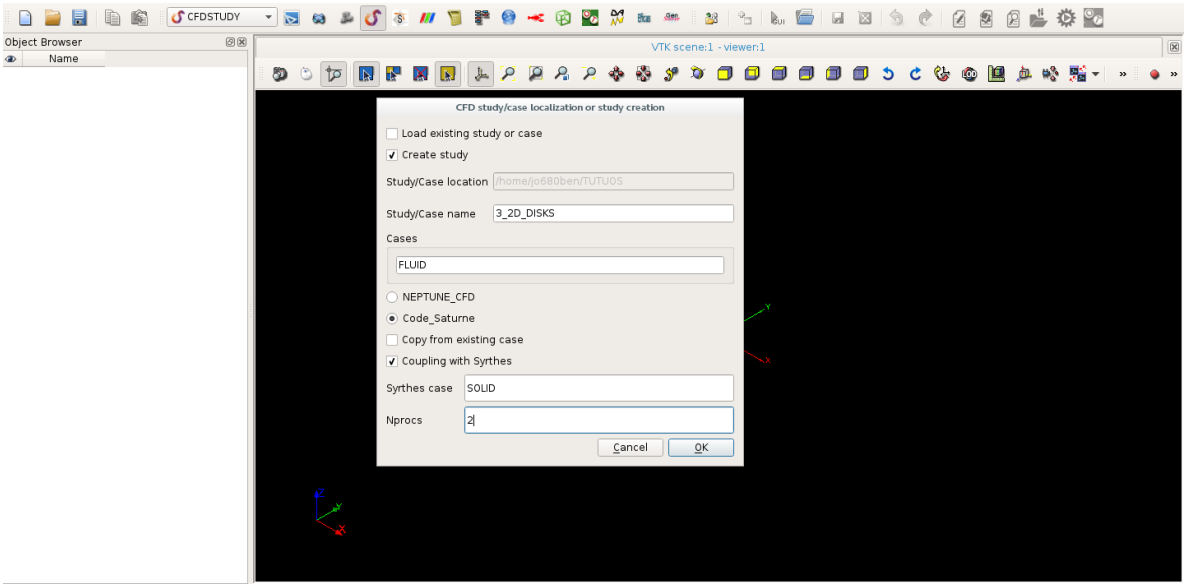

Figure II.1: Coupled study creation with SALOME module CFDSTUDY.

At this step, it is possible to choose the total number of processes that will be used for a coupled computation. Here 2 is chosen.

- Create coupled study in a terminal with the following command line:

```
$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
```

Second, mesh files have to be copied in the study. The fluid mesh has to be copied in the subdirectory 🗀`MESH`. The solid mesh must be copied in the subdirectory 🗀`SOLID`.

Third, to set up the solid settings, launch SYRTHES Graphical User Interface (GUI) in SALOME by

opening SYRTHES module[1] or in a terminal by typing `$ syrthes.gui &` inside the subdirectory SOLID.

Fourth, to set up the fluid settings, launch the *Code_Saturne* GUI in SALOME or in a terminal inside the subdirectory FLUID/DATA (script SaturneGUI) for the fluid computation alone.

1.3 Geometry

As said above, the simplified configuration represents a 2D cross section of the electric wires inside the sheath. The 3 wires inside the sheath are represented as 3 disks inside a larger ring and it is assumed that the 3 disks are in contact with an air flow inside the electric sheath.

The geometry is shown on figure II.2.

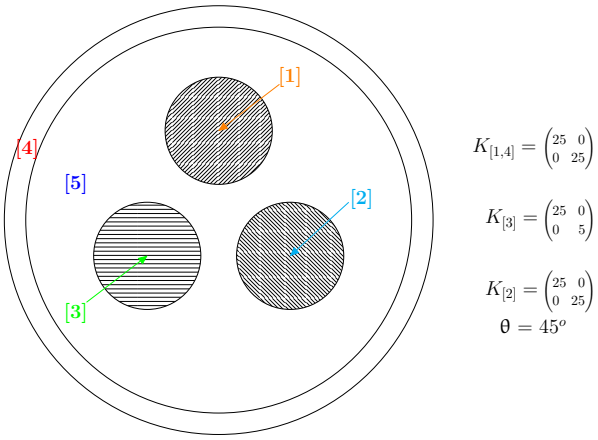

Figure II.2: Geometry with the solid domain (parts $1, 2, 3, 4$) and the fluid domain (part 5).

1.4 Meshes characteristics

- **Mesh of the solid domain:**

The mesh of the solid part contains 11688 nodes (P_1 discretization) and 5688 elements. Materials properties and boundary conditions are specified in the next part using the references given on figure II.3.

Type: unstructured mesh
Mesh generator used: SIMAIL

- **Mesh of the fluid domain:**

The fluid mesh contains 3866 nodes. **Mesh Quality Criteria** run type in the *Code_Saturne* Graphical User Interface (**Calculation management**, **Pepare batch calculation** section) can be used to check the quality of the mesh and to help to identify the references associated to the boundary conditions (1 is used for all solid boundaries, 2 for the front fluid face, and 3 for the back fluid face). See figure II.4.

Type: unstructured mesh
Mesh generator used: SIMAIL

[1] Once a .syd file is present, you can right-click on it in CFDSTUDY object browser and launch SYRTHES GUI from the contextual menu.

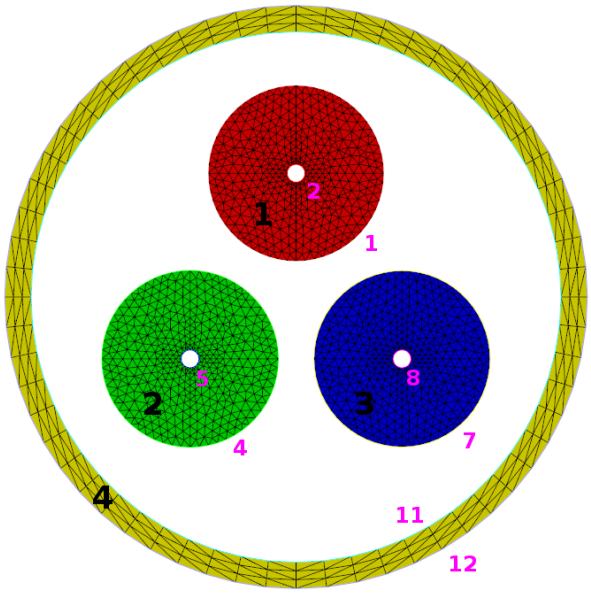

Figure II.3: References for the solid interior zones (black) and solid boundary zones (pink).

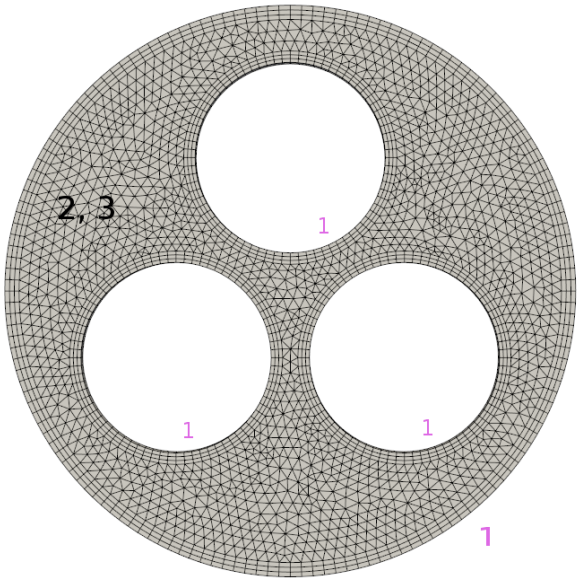

Figure II.4: fluid boundary faces references

1.5 Data settings for solid and fluid domains

- **Solid domain:**

Material properties for the solid domain are given by part (1 to 3 for the electric wires and 4 for the disk for the electric sheath) in the subsequent table:

	Conductivity type	**Values** (W/m/°C)	**Volume reference**
Disk 1	Isotropic	$k_{11} = 25$	1
Disk 2	Orthotropic	$k_{11} = 25$; $k_{22} = 5$	2
Disk 3	Anisotropic	$k_{11} = 25$; $k_{22} = 5$ $\alpha = 45^o$	3
Disk 4	Isotropic	$k_{11} = 25$	4

Physical properties	**Values**
Density $[\rho]$	7700 (kg/m^2)
Specific heat $[C_p]$	460 $(J/kg/m^3)$

Initial and boundary conditions for the solid domain should be defined as follows:

Initial conditions	
Temperature condition	$T_{ini,s} = 20$°C

Boundary conditions	**Value**	**Surface reference**
Heat exchange conditions $(q_{w,ext})$	$T_{ext} = 90$°C. ; $h_{ext} = 1000(W/m^2.K)$	2 or 5 or 8

For the coupled case, nothing needs to be changed in this part.

- **Fluid domain:**

Some characteristics of the air flow that should be checked in the GUI are given hereafter:

Modeling feature	**choice**
Time step	constant in time and uniform in space
Turbulence model	$k - \varepsilon$ LP
Thermal model	Temperature (°C)

In the air flow, density is a function of the temperature and gravity force is taken into account. The 3 disks, which are warmer than the air flow, generate a temperature difference creating a fluid movement. The warmer air flow is moving to the top and the colder air flow to the bottom of the fluid domain.

The density follows an ideal gas law that can be specified in the GUI:

$$\rho = \frac{p_0}{R_g \ (T + 273.15)} \tag{II.1}$$

where ρ is the density, T is the temperature (°C), $R_g = 287$ $(m^2.s^{-2}.K^{-1})$ the ideal gas constant and $p_0 = 101325$ (Pa) the reference pressure (atmospheric pressure).

Initial conditions are defined below:

Initial conditions	
Temperature condition	$T_{ini,f} = 20$°C.

Symmetry conditions are imposed on the front and back face, and walls are set on all remaining faces (those coupling fluid and solid domain).

Boundary conditions	Values	Surface reference
Walls (Heat exchange $q_{w,ext}$)	T_{ext}(°C) ; $h_{ext}(W/m^2.K)$	1
Symmetry		2 or 3

For the fluid computation alone $T_{ext} = 30$°C and $h_{ext} = 10W/m^2.K$ while for the coupled case, simply set $T_{ext} = 0$°C and $h_{ext} = 0W/m^2.K$.

2 Computations of the 3 2D disks configuration

If not already done, set all data necessary to run the fluid and solid part configurations separately based on the description of the previous section. All the parameters necessary to this study can be defined through *Code_Saturne* GUI and SYRTHES GUI respectively.

2.1 Time stepping parameters

Set now all the time stepping parameters on both sides: *Code_Saturne* and SYRTHES.

Time stepping parameters of solid computation	
Reference time step	10 (s)
Number of iterations	100
Time stepping parameters of fluid computation	
Reference time step	0.1 (s)
Number of iterations	100

These time stepping parameters will be set to run the fluid and solid computations **independently from one another**.

2.2 Output management

Standard options for output management will be used. Only one monitoring point will be created for the solid conduction computation at the following coordinates:

Probe	x (m)	y (m)
1	0.003	-1.2

The output frequency for this probe can be every 10 time steps.

The temperature field can be saved every 25 time steps.

2.3 Results for the uncoupled cases

The velocity and **temperature** field can be post-processed with ParaVis.

As examples, Figure II.5 shows the evolution of the temperature in the solid domain and Figure II.6 the evolution of the temperature in the fluid domain without **Conjugate heat transfer**.

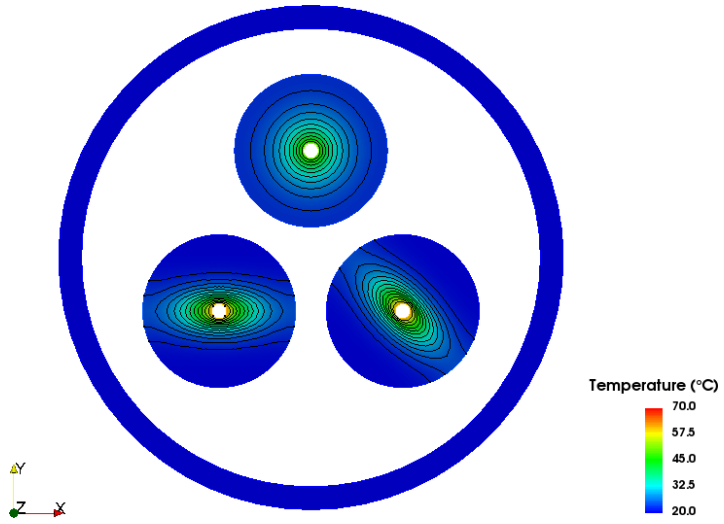

Figure II.5: The temperature evolution in the **solid domain without coupling method**

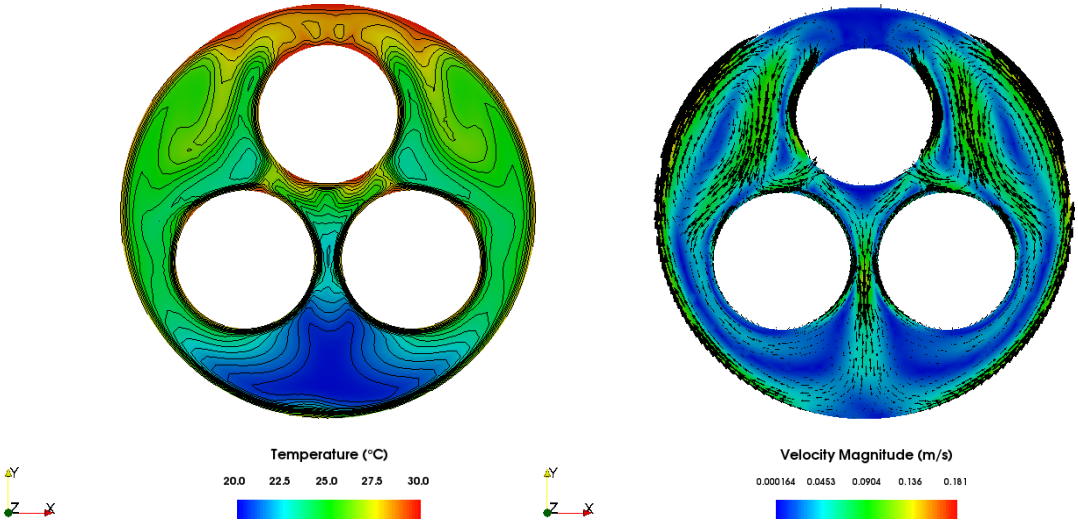

Figure II.6: The temperature evolution in the **fluid domain without coupling method**

2.4 Parameters for the coupled computation

Conjugate heat transfer has to be enabled and the coupling interfaces specified on both sides.

Then time stepping parameters have to be modified to be able to see the effect of the conjugate heat transfer phenomenon between the solid and fluid domains. For this reason, we increase the number of iterations and the reference time step for the fluid and the solid part[2].

[2]By default, the smallest number of iterations will be used to drive the coupling computation. If we choose a number of iterations of 10000 for the fluid domain and 5000 for the solid domain, the coupling computation will be stopped after 5000 instead of 10000.

Time stepping parameters of solid computation	
Reference time step	0.5 (s)
Number of iterations	600

Time stepping parameters of fluid computation	
Reference time step	0.5 (s)
Number of iterations	600

The **Improved pressure interpolation in stratified flow** option has to be checked in *Code_Saturne* GUI (**Numerical parameters**, **Global parameters** section).

2.5 Results for the coupled computation

As an example, Figure II.7 shows the evolution of the temperature in the solid and fluid area with the **conjugate heat transfer** enabled. The natural convection in the fluid domain due to the temperature difference imposed by the solid disks is clearly visible with the velocity displayed as a vector field.

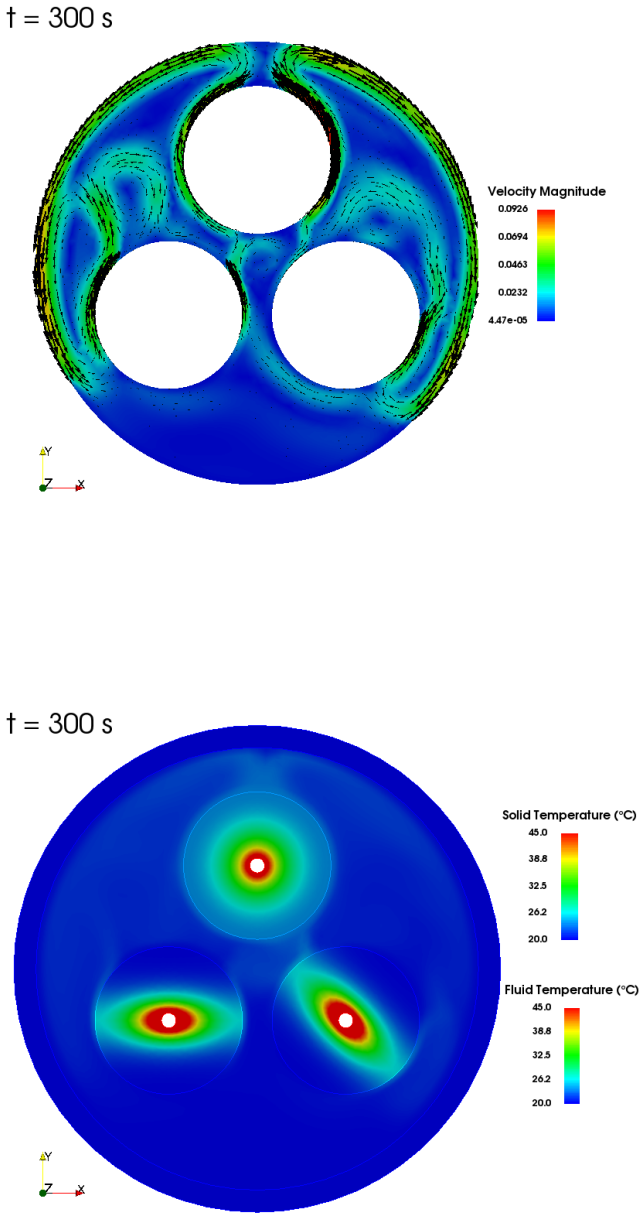

Figure II.7: Evolution of temperature and velocity magnitude

Part III

Step by step description

1 Detailed tutorial step by step

1.1 Creation of the study in a terminal

• **Step 1**: check the post-install required for coupling *Code_Saturne* with SYRTHES.
The first step is to check the post-install required for coupling with SYRTHES and verify if the SYRTHES •PATH is correctly known in the system environment. We just need to edit the batch file[1] name `code_saturne.cfg` as below:

```
$ vim <install-prefix>/etc/code_saturne.cfg
>### Set the location to the SYRTHES installation directory.
> syrthes = <install-prefix-syrthes>
```

• **Step 2**: source the `syrthes.profile` file in your user environment.
Before using SYRTHES alone, you have to copy and source this file to define SYRTHES environment variables (like `$SYRTHES4_HOME`) in your terminal, as follows:

```
$ cp <install-prefix-syrthes>/bin/syrthes.profile .
$ source syrthes.profile
$ echo $SYRTHES4_HOME (to check the SYRTHES PATH in your environment)
```

After having defined correctly your environment, to be able to launch a coupling computation *Code_Saturne*-SYRTHES or a SYRTHES computation alone, you just have to create the coupling study directory.

• **Step 3**: create the 🗀`3_2D_DISKS` study directory, and the two case subdirectories 🗀`FLUID` and 🗀`SOLID`.

This is done using the standard command:

```
$ code_saturne create -s 3_2D_DISKS -c FLUID --syrthes SOLID
> code_saturne 3.0 study/case generation
> o Creating study '3_2D_DISKS' ...
> o Creating case 'FLUID' ...
> SYRTHES4 home directory:  <install-prefix-syrthes>
> MPI home directory:  /usr
>
>************************************************
> SOLID : creating SYRTHES case ...
> <install-prefix-syrthes>
> OK !
>************************************************
```

• **Remark**: The fluid mesh must be copied in the directory 🗀`MESH`. The solid mesh must be copied in the subdirectory 🗀`SOLID`.

[1] see the installation guide, name `install.pdf`, in `<install-prefix>/share/doc/code_saturne/` directory.

1.2 Preparing and launching SYRTHES computation alone

Preparation of SYRTHES computation alone can be one in 6 steps:

- **Step 1**: Launch SYRTHES module in SALOME or SYRTHES GUI in a terminal,
- **Step 2**: Create a **New Data File**,
- **Step 3**: Check the name of the mesh and convert this one in .syr format,
- **Step 4**: Define the initial and boundary conditions for the conduction problem,
- **Step 5**: Define the physical properties of each disk {1, 2, 3 and 4},
- **Step 6**: Running the SYRTHES computation alone.

• **Step 1**: launch SYRTHES module in SALOME or SYRTHES GUI in a terminal.
SYRTHES GUI can be launched by the following command lines in the solid subdirectory:

```
$ cd 3_2D_DISKS/SOLID/
$ syrthes.gui &
```

• **Step 2**: choose `New Data File` inside the pop-up window.

Figure III.1: Opening pop-up window for SYRTHES GUI in *SALOME*

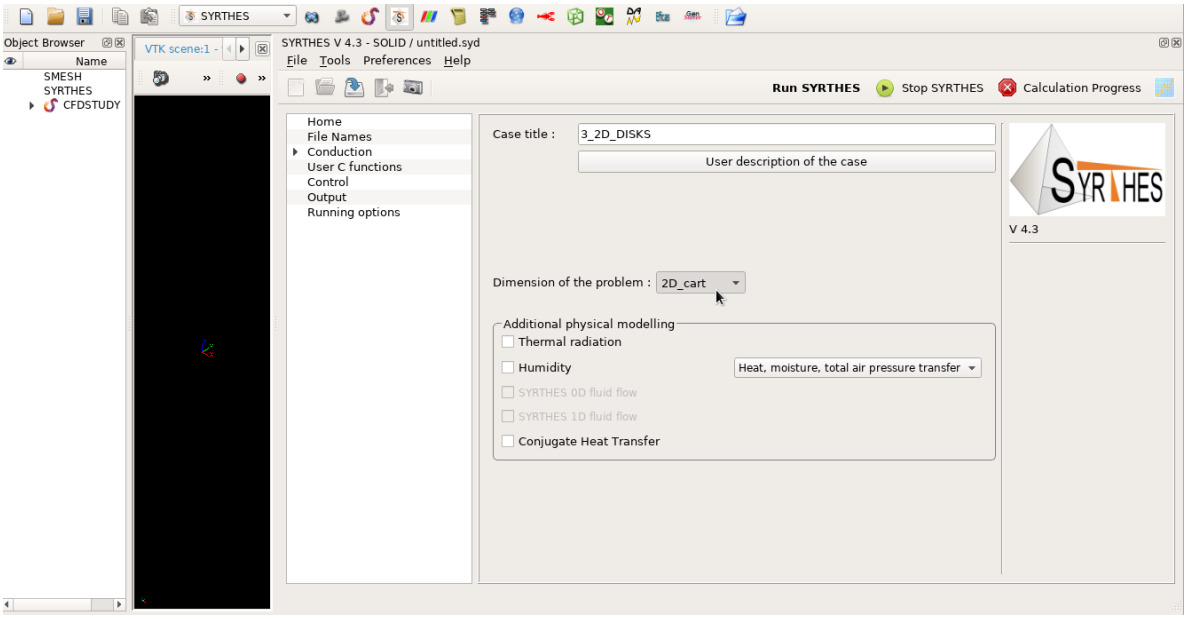

Figure III.2: Define the dimension and physical modelling of the treated problem

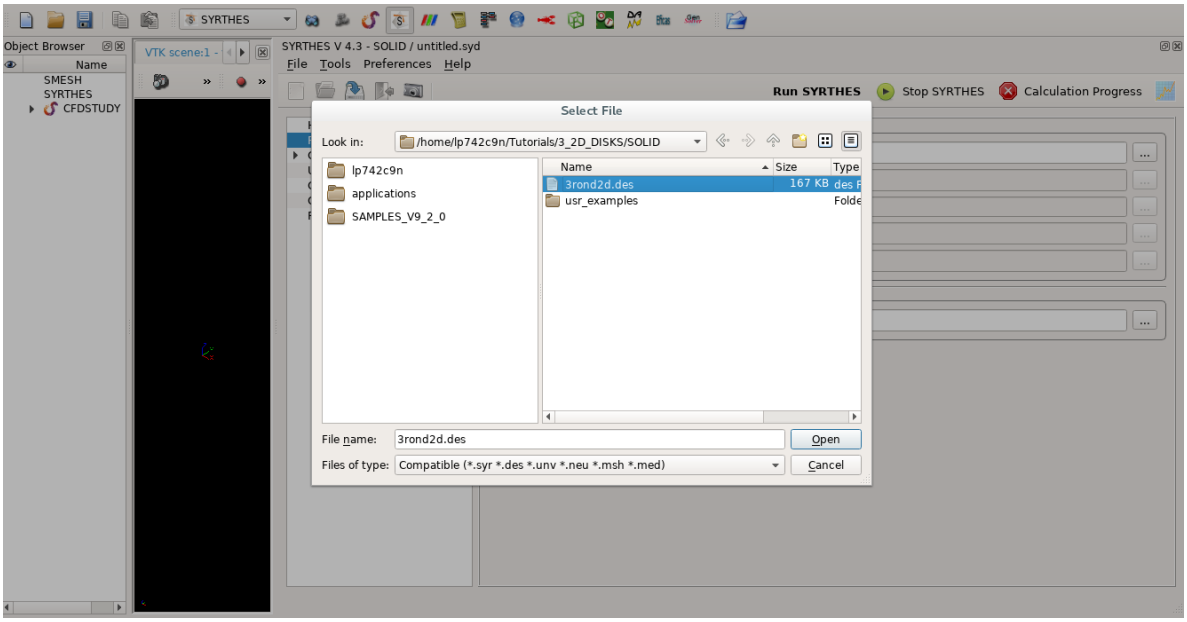

Figure III.3: Choose the 2D solid mesh file with the format `.des`.

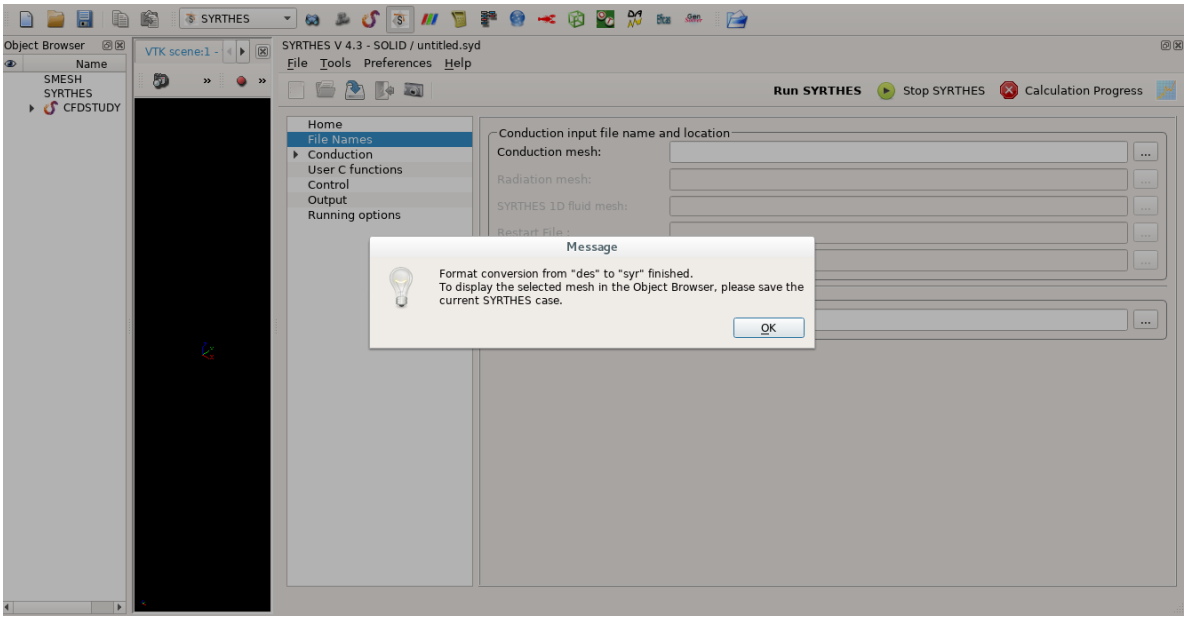

Figure III.4: The SYRTHES (GUI) directly converts the `.des` to the `.syr` format.

• **Remark**: Inside SYRTHES GUI, we can load the SIMAIL format `*.des` for the solid mesh. This one will be automatically transformed to the `*.syr` format.
It can also be done with the following command line:

```
$ convert2syrthes4 -m 3rond2d.des
```

• **Remark**: You can convert the `*.syr` format into a `*.med` format. Like that, you can load the `*.med` file inside SALOME, after having used this command line below:

```
$ syrthes4med30 -m 3rond2d.syr -o 3rond2d.med
```

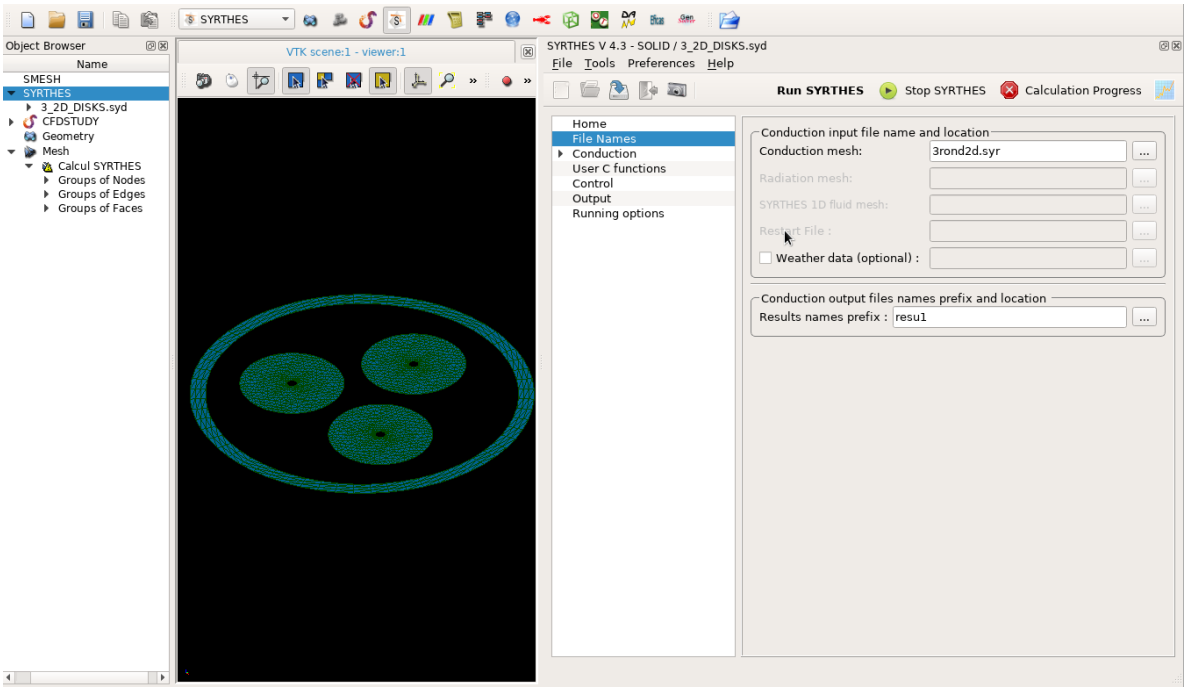

Figure III.5: Choose a name for the results files `.res`, `.his` and `.rdt`

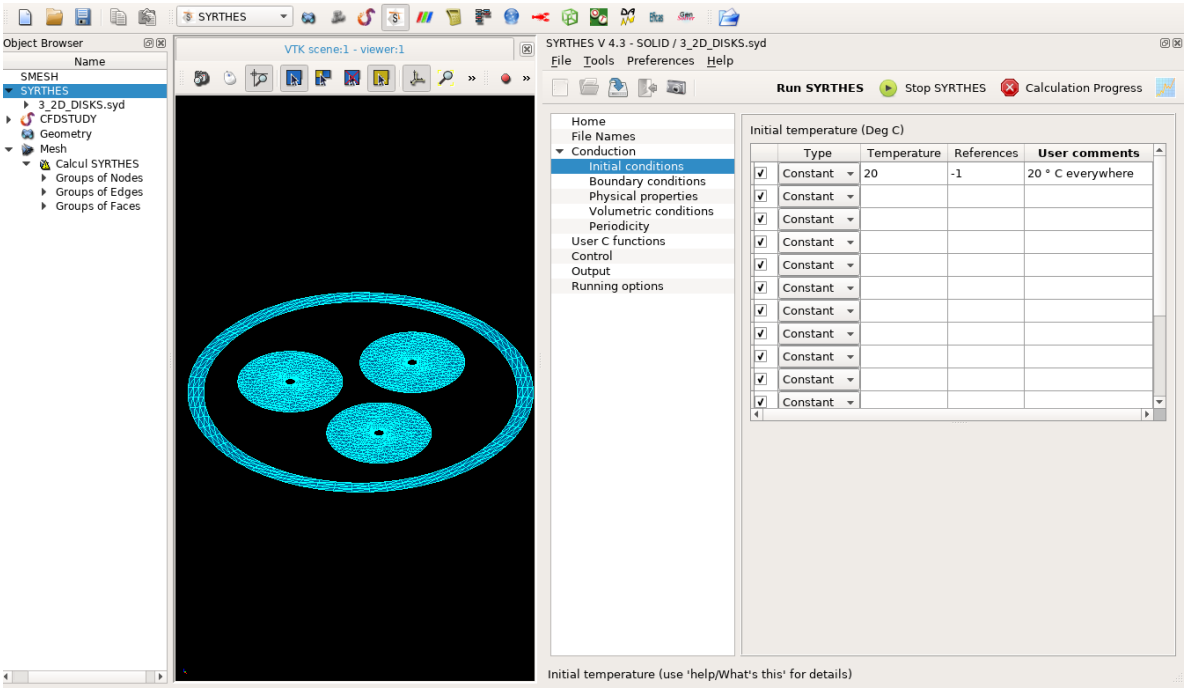

Figure III.6: Define the initial temperature conditions inside the different disks.

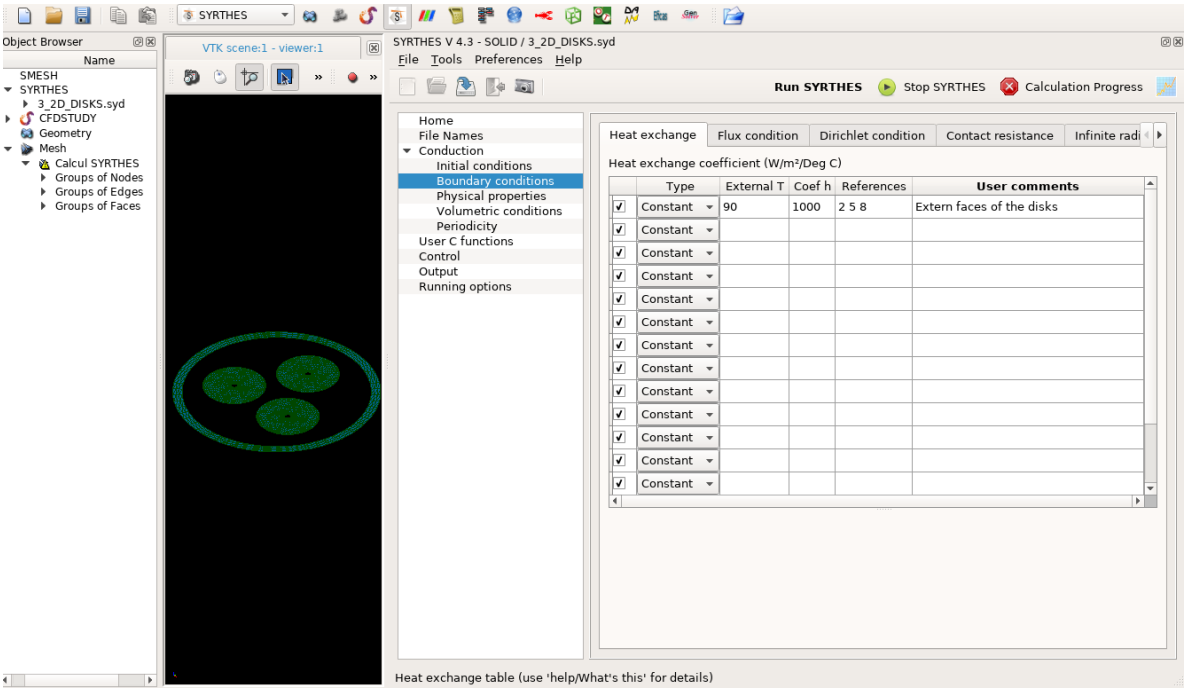

Figure III.7: Define the temperature boundary conditions for the extern faces of the three disks.

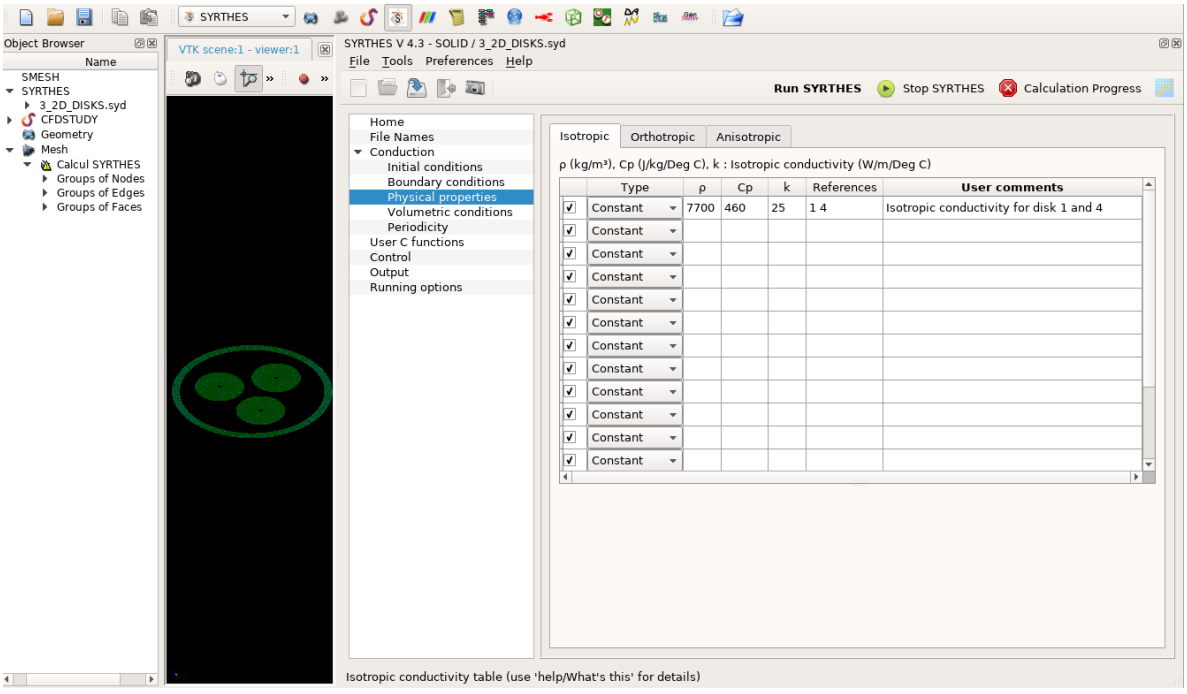

Figure III.8: Define the physical properties for the disk 1 and 4 with isotropic conductivity.

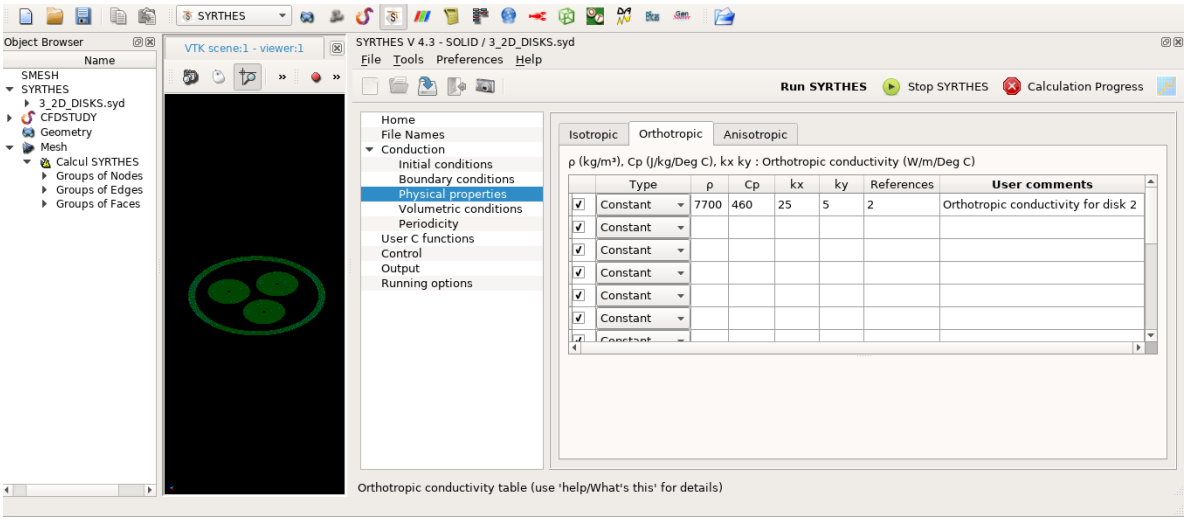

Figure III.9: Define the physical properties for the disk 2 with orthotropic conductivity.

• **Remark**: To correctly identify the volume references associated to a specific physical property, we can check the mesh regions directly inside ParaVis.

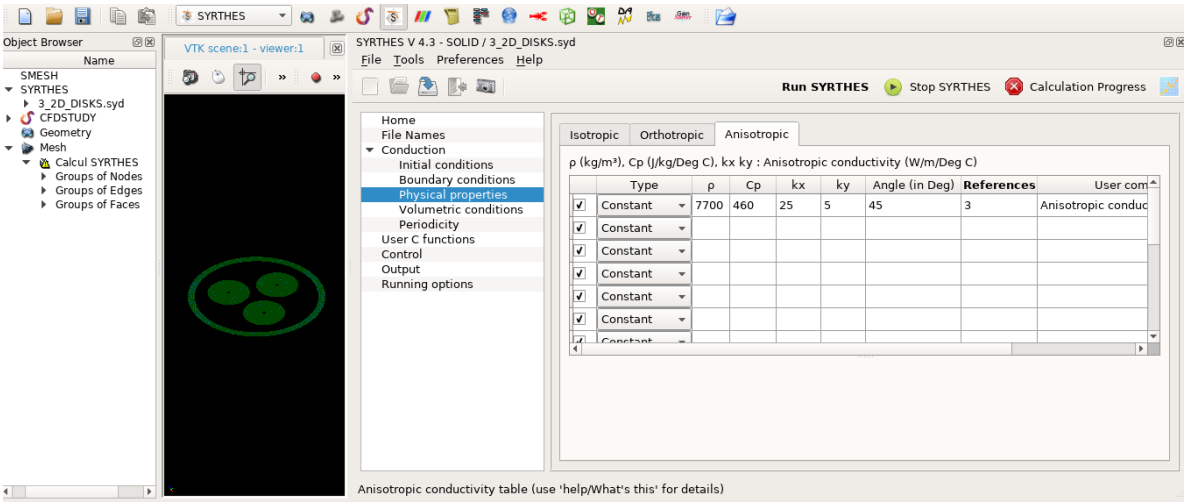

Figure III.10: Define the Physical properties for the disk 3 with anisotropic conductivity.

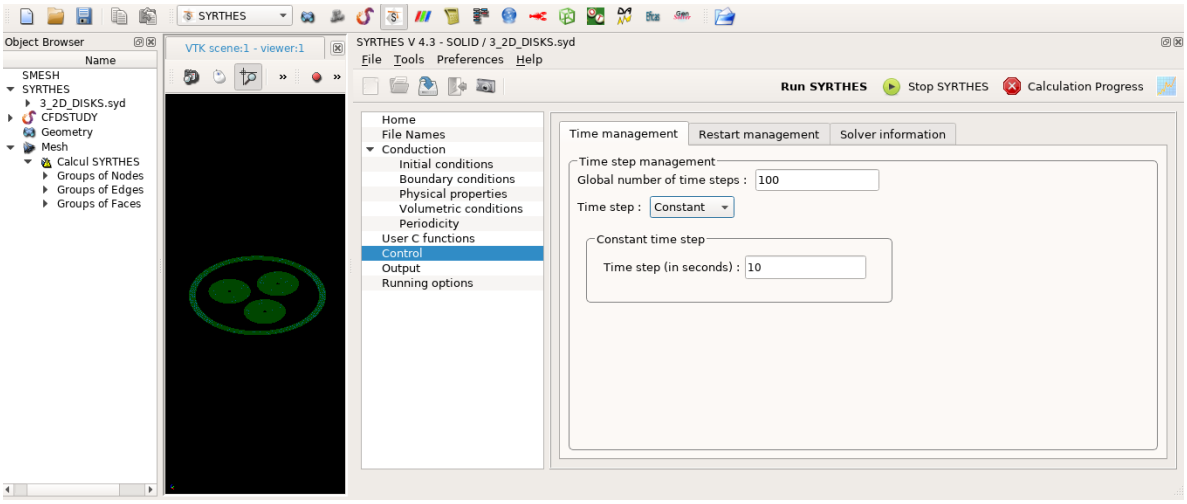

Figure III.11: Define the global number of time steps and the time step for the 2D solid conduction computation.

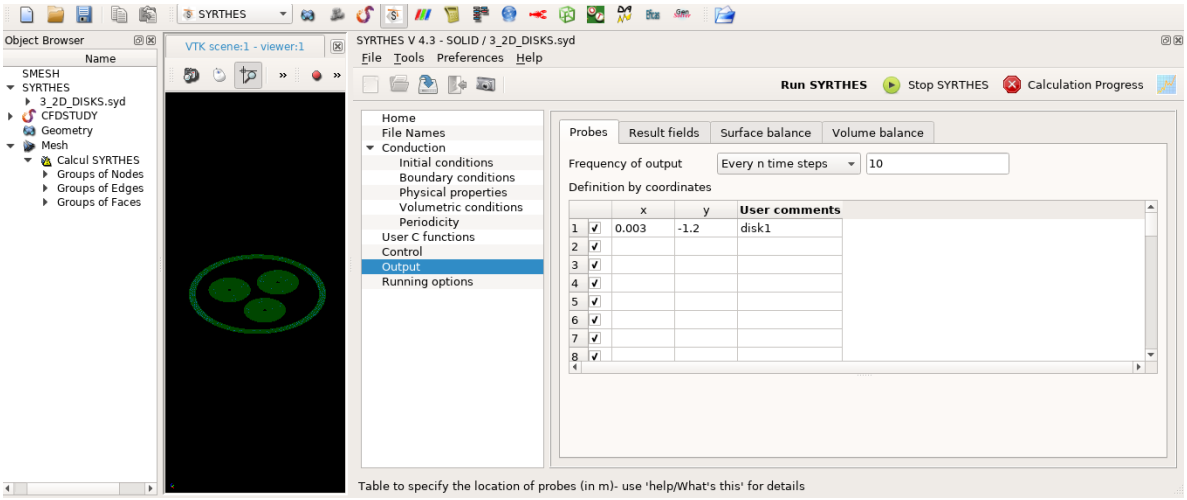

Figure III.12: Define the probe coordinates for output management.

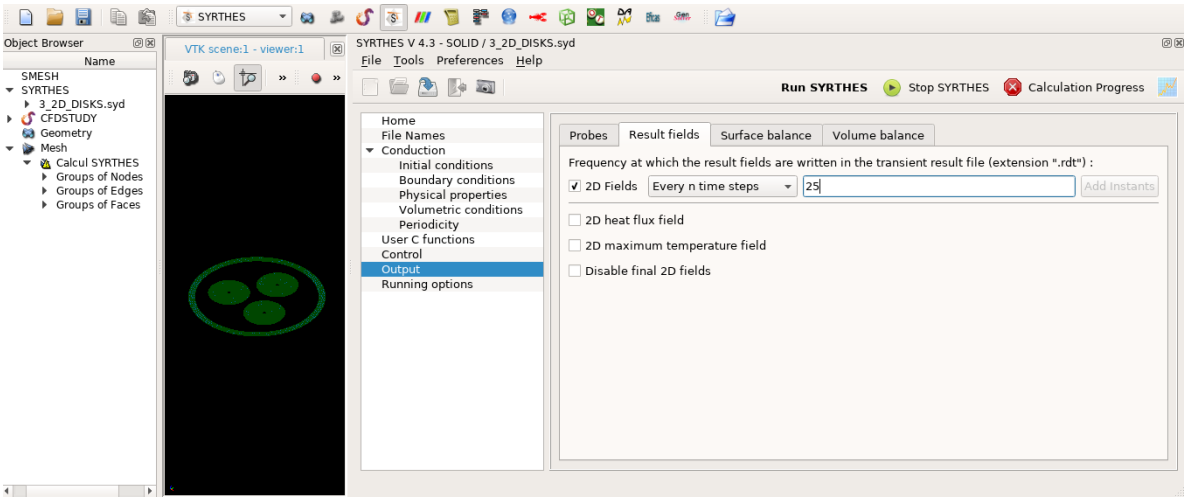

Figure III.13: Define the frequency at which the results fields are written

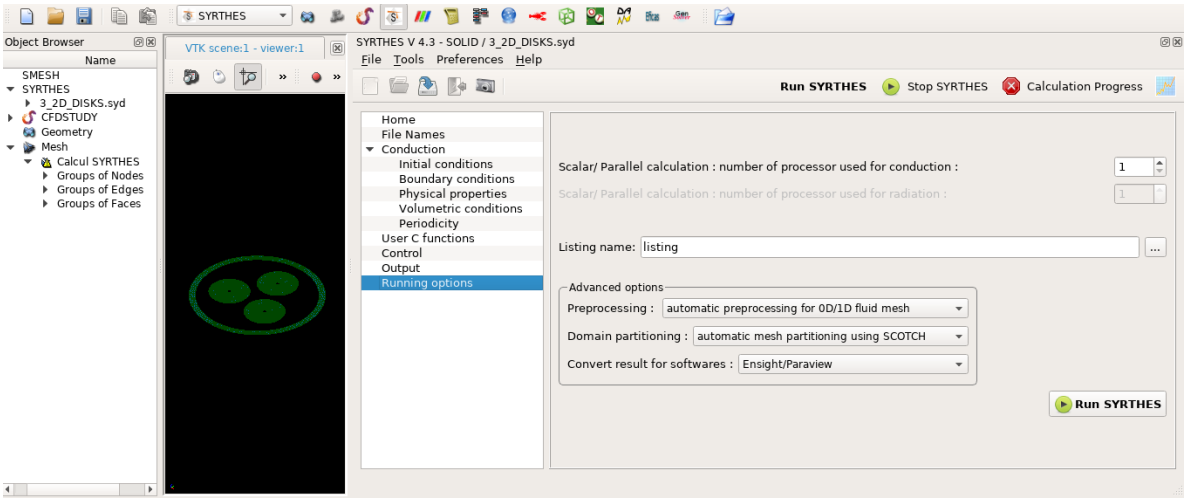

Figure III.14: Define the file name of the SYRTHES listing and the number of processors used.

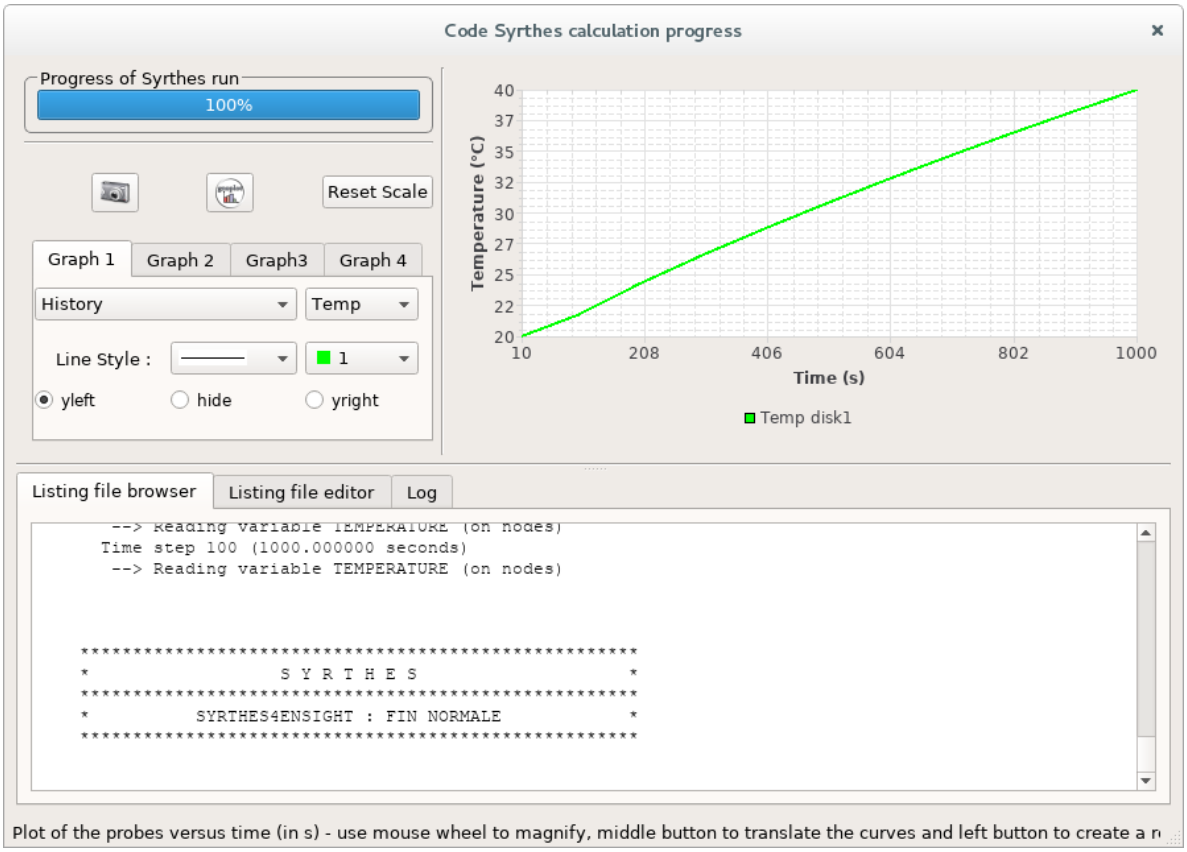

Figure III.15: Screenshot of the computation progress window (the graph represents the value of the temperature at the monitoring point).

1.3 Preparing and launching the *Code_Saturne* computation alone

The main steps of the preparation of the fluid computation alone can be the following ones:
• **Step 1**: Launch *Code_Saturne* GUI from the object browser or the tool bar in SALOME module CFDSTUDY (or `./SaturneGUI` in command line),
• **Step 2**: Create a **New case**,
• **Step 3**: Check the quality of the fluid mesh by running a **Mesh quality criteria** calculation,
• **Step 4**: Define the physical properties of the disk for the air flow,
• **Step 5**: Define the initial and boundary conditions for the air flow problem,
• **Step 6**: Run the *Code_Saturne* computation alone.

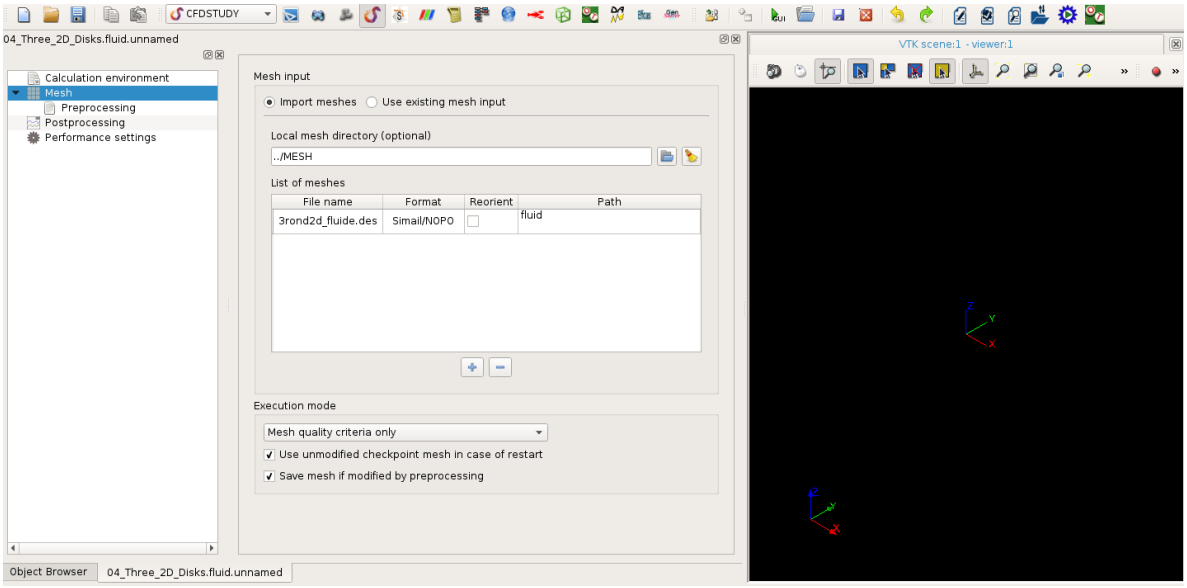

Figure III.16: Choose the fluid mesh with *Code_Saturne* (GUI) and select the execution mode *Mesh quality criteria only*

Run a calculation to check the mesh (the *preprocessor.log* file will be used later). Then select the option *Standart computation* as execution mode and set up physical properties as follow.

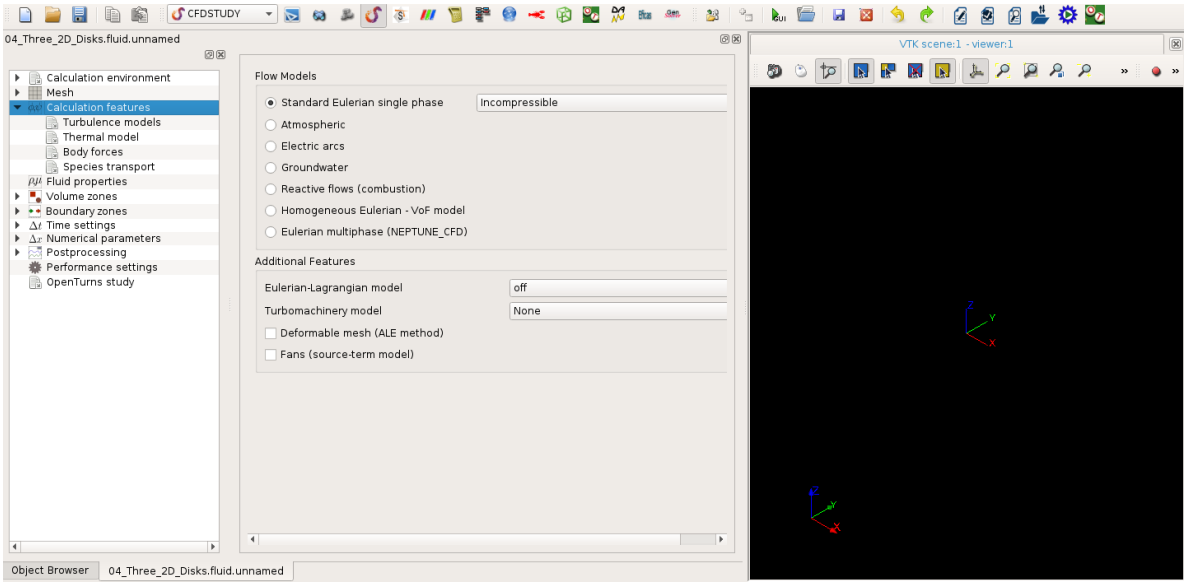

Figure III.17: Define the calculation features adapted to the air flow inside the fluid domain.

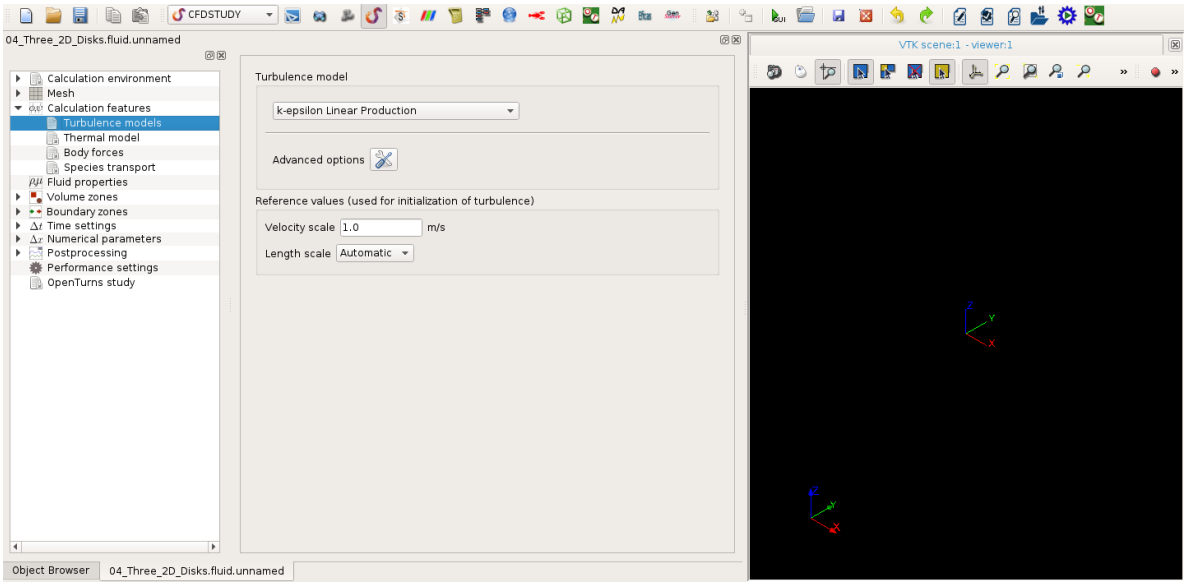

Figure III.18: Select the $k-\varepsilon$ LP as turbulence model

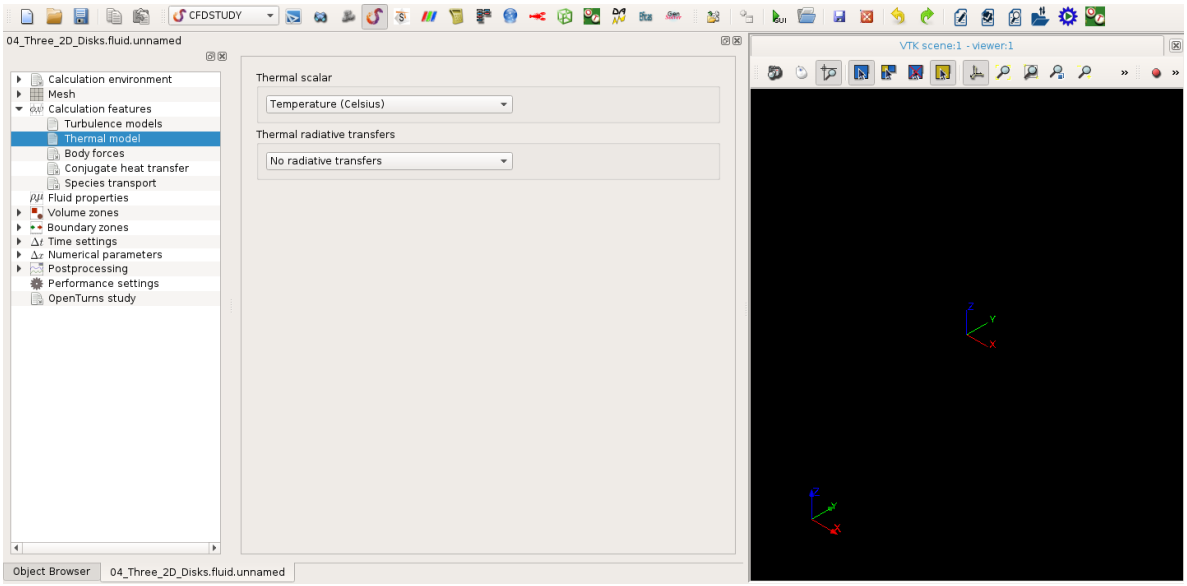

Figure III.19: Choose the Temperature scalar.

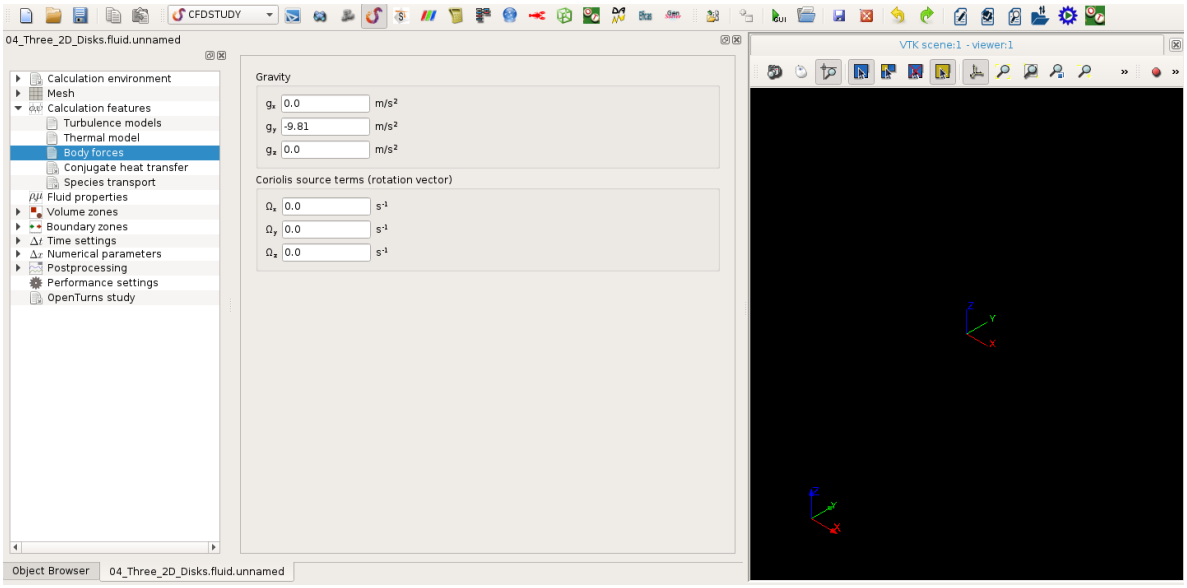

Figure III.20: Define the gravity

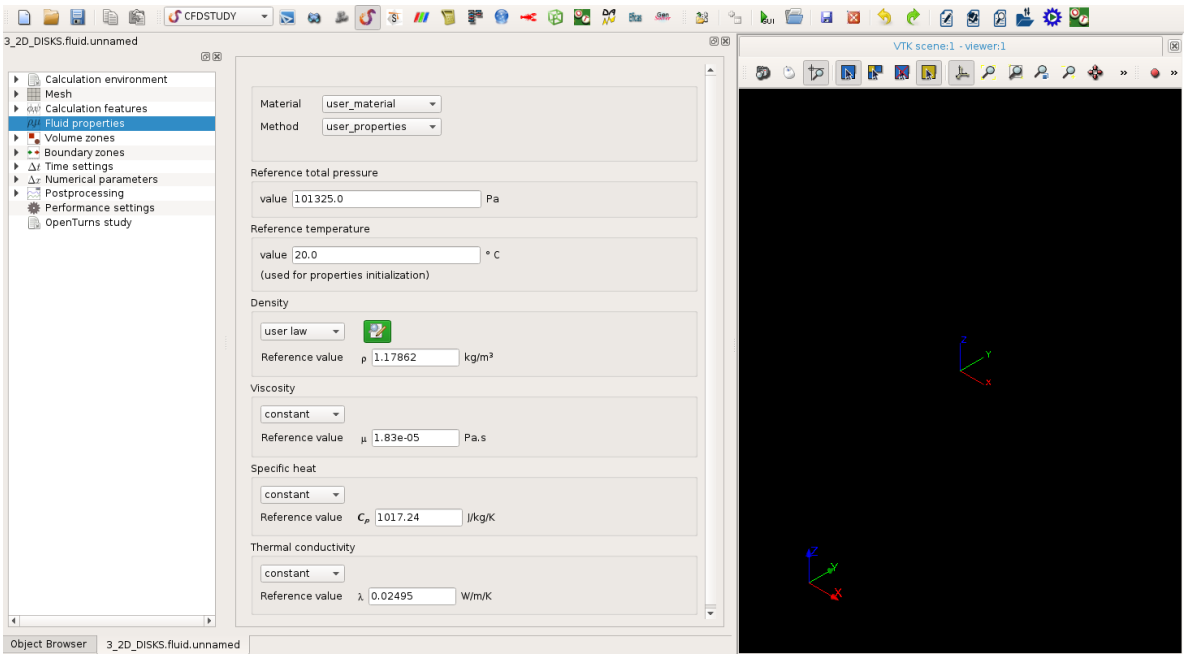

Figure III.21: Define the physical properties.

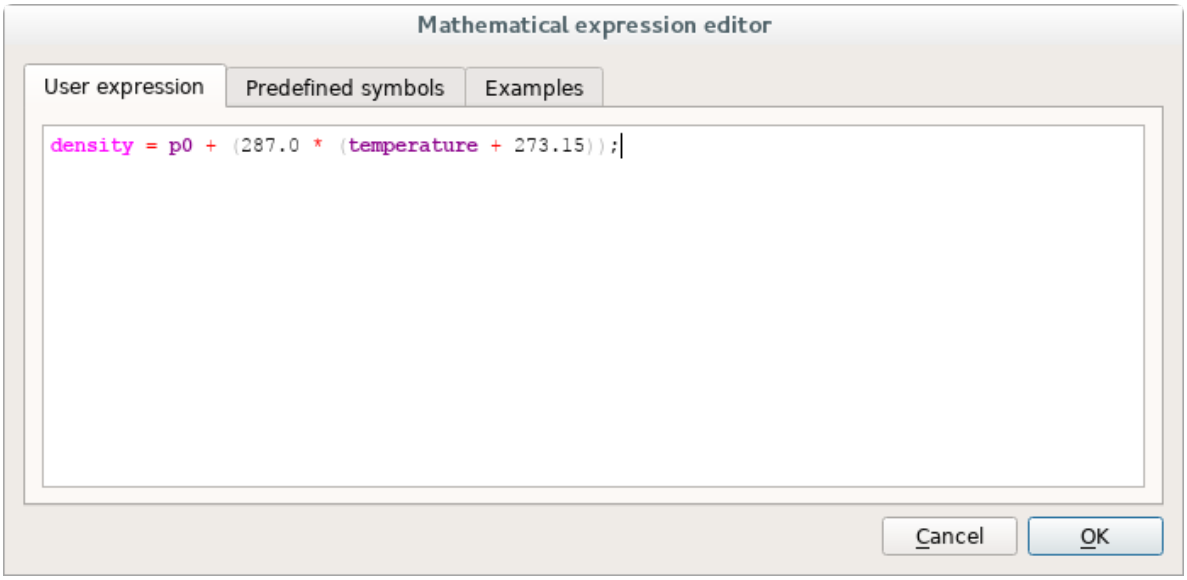

Figure III.22: Define the variable density with an ideal gas law.

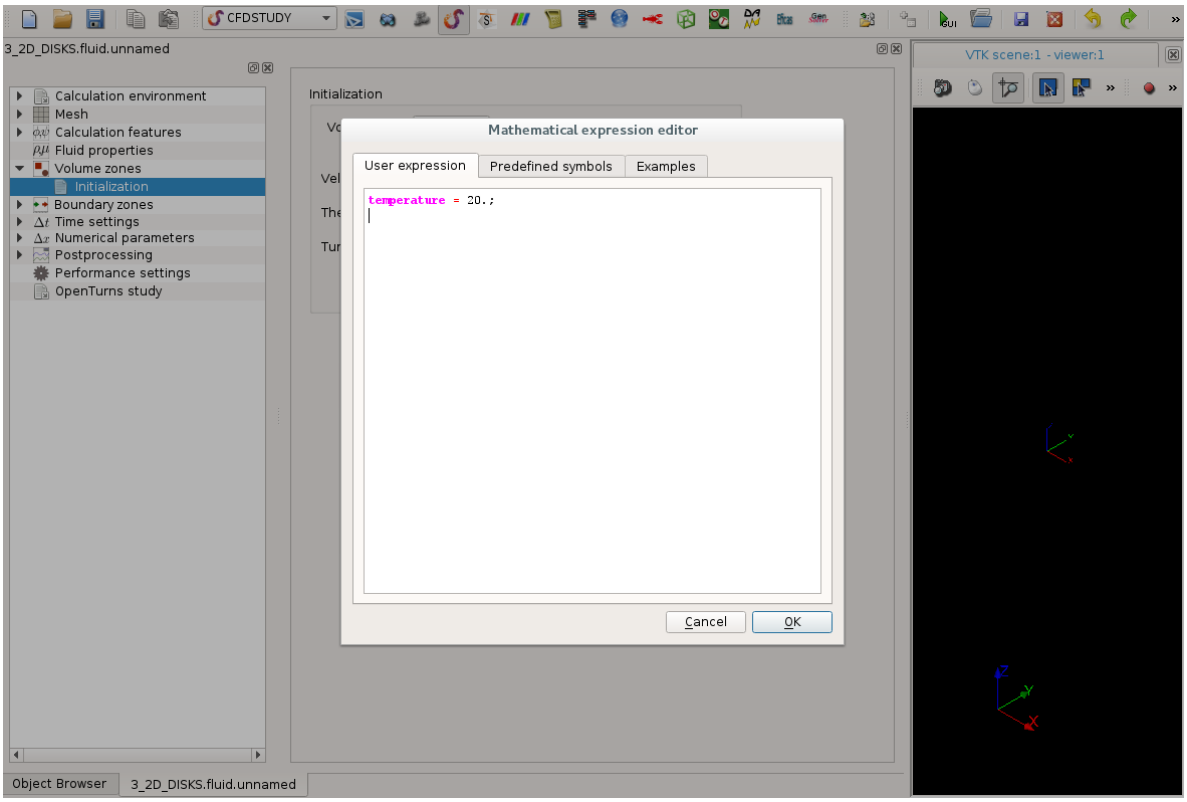

Figure III.23: Initialization of the temperature.

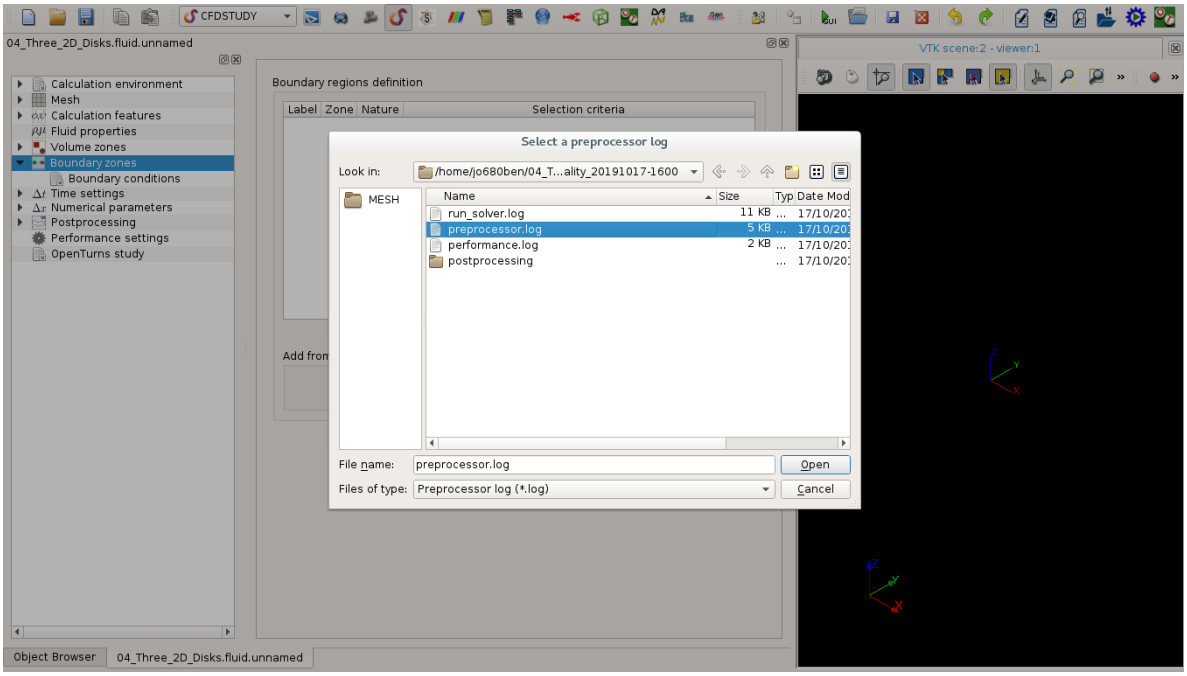

Figure III.24: Load the `preprocessor.log` file inside the *Code_Saturne* (GUI) to define boundary regions.

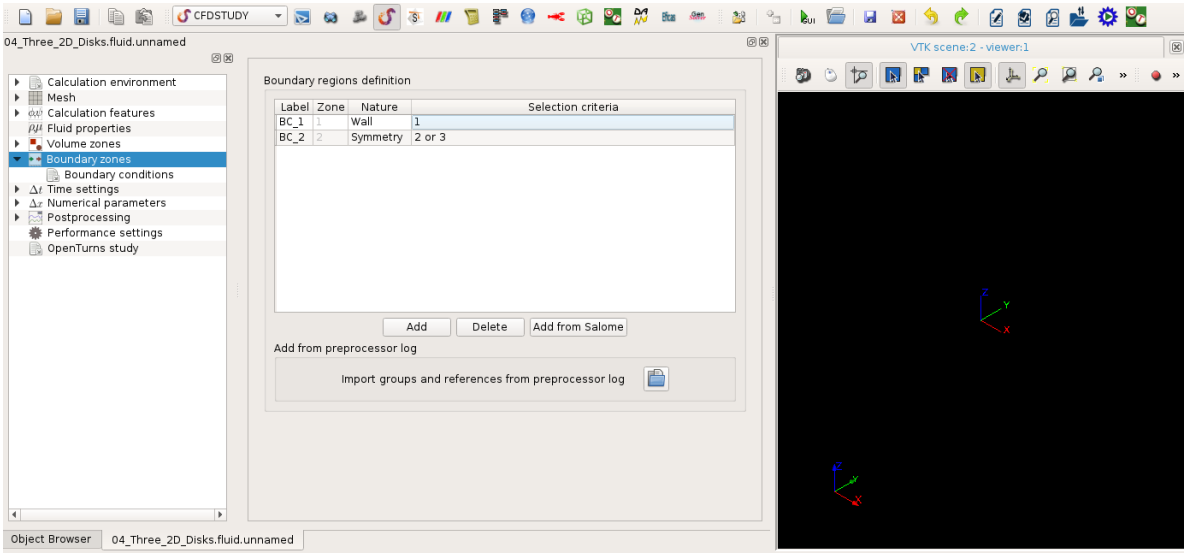

Figure III.25: Once the boundary regions automatically loaded, define the boundary conditions.

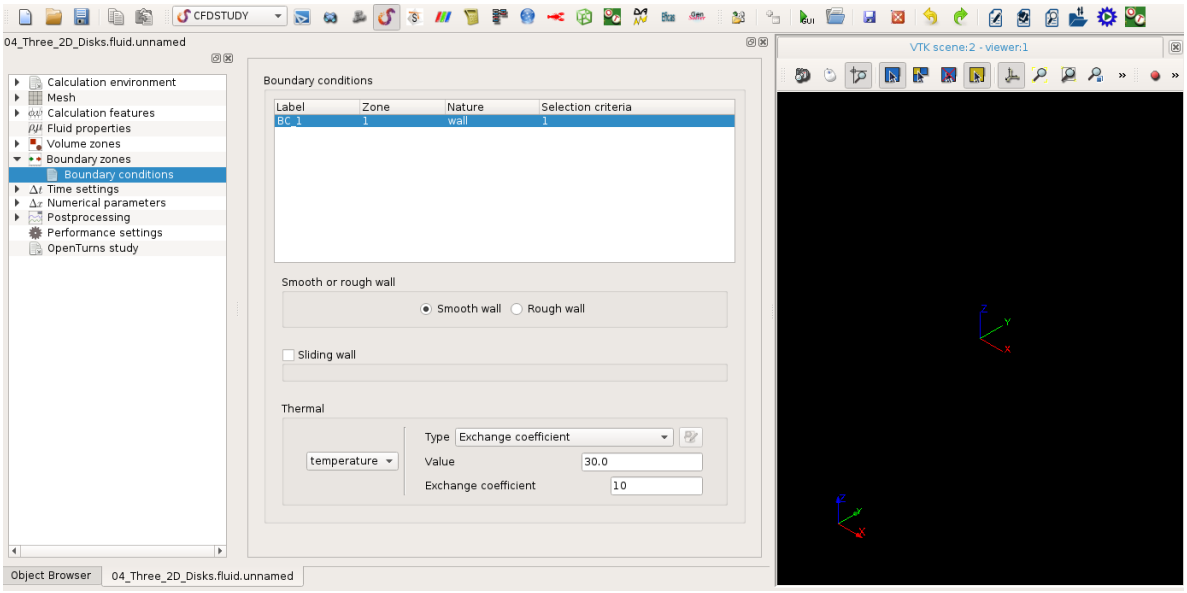

Figure III.26: Define a thermal transfer condition as wall boundary condition with an external wall temperature $T_{ext} = 30°C$ and an exchange coefficient $q_{ext} = 10\ (W/m^2.K)$.

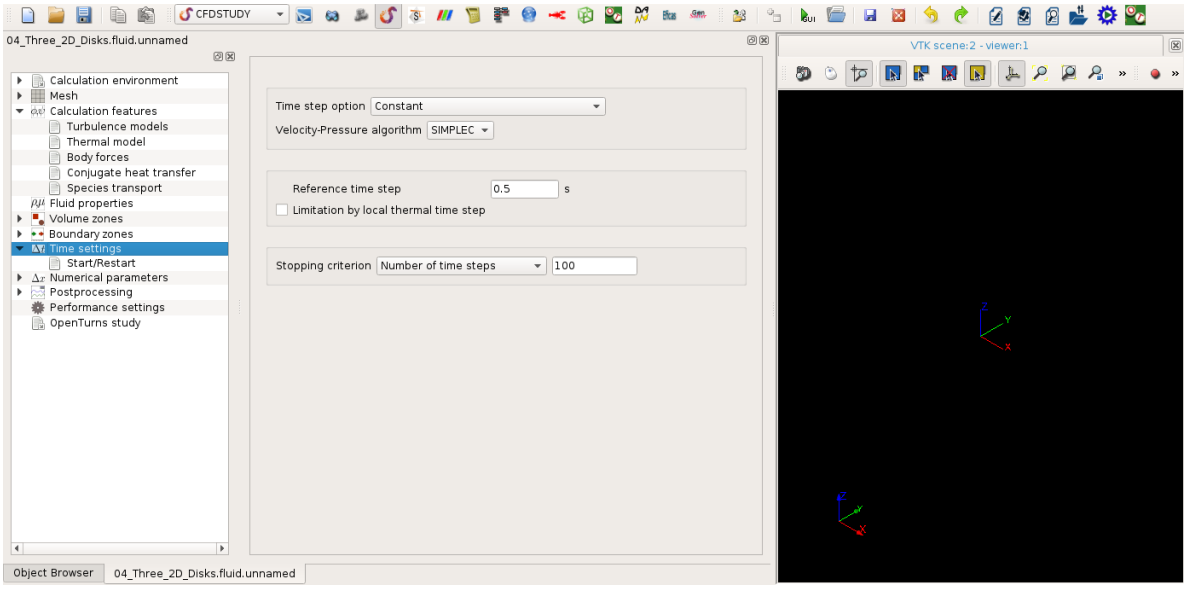

Figure III.27: Define the iterations number and time step.

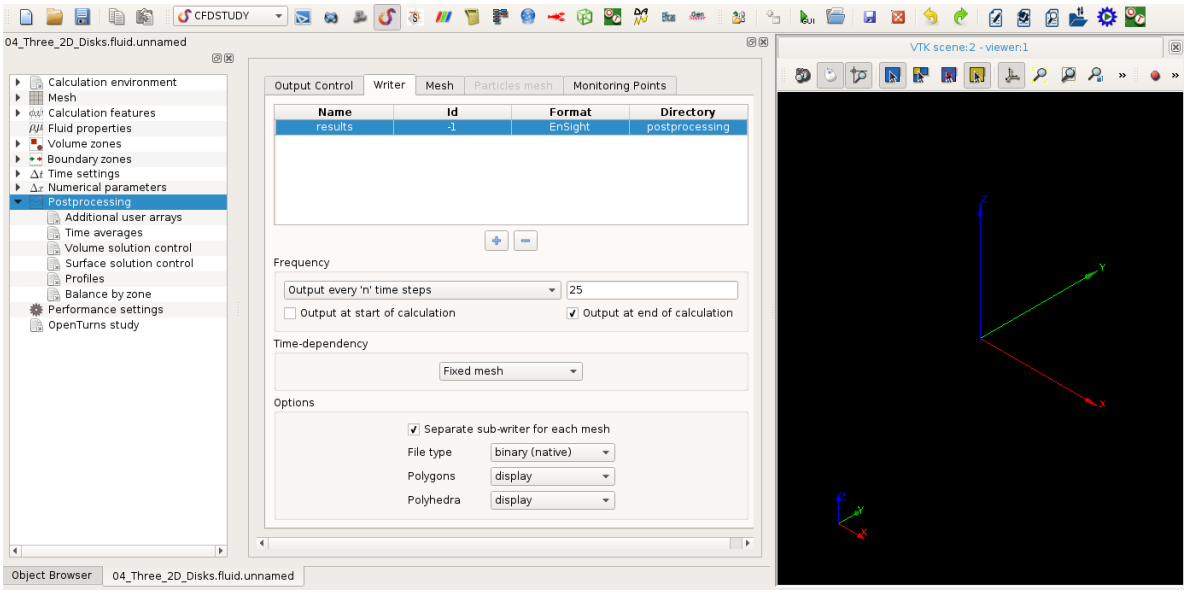

Figure III.28: Define the writer and frequency output inside the *Code_Saturne* (GUI).

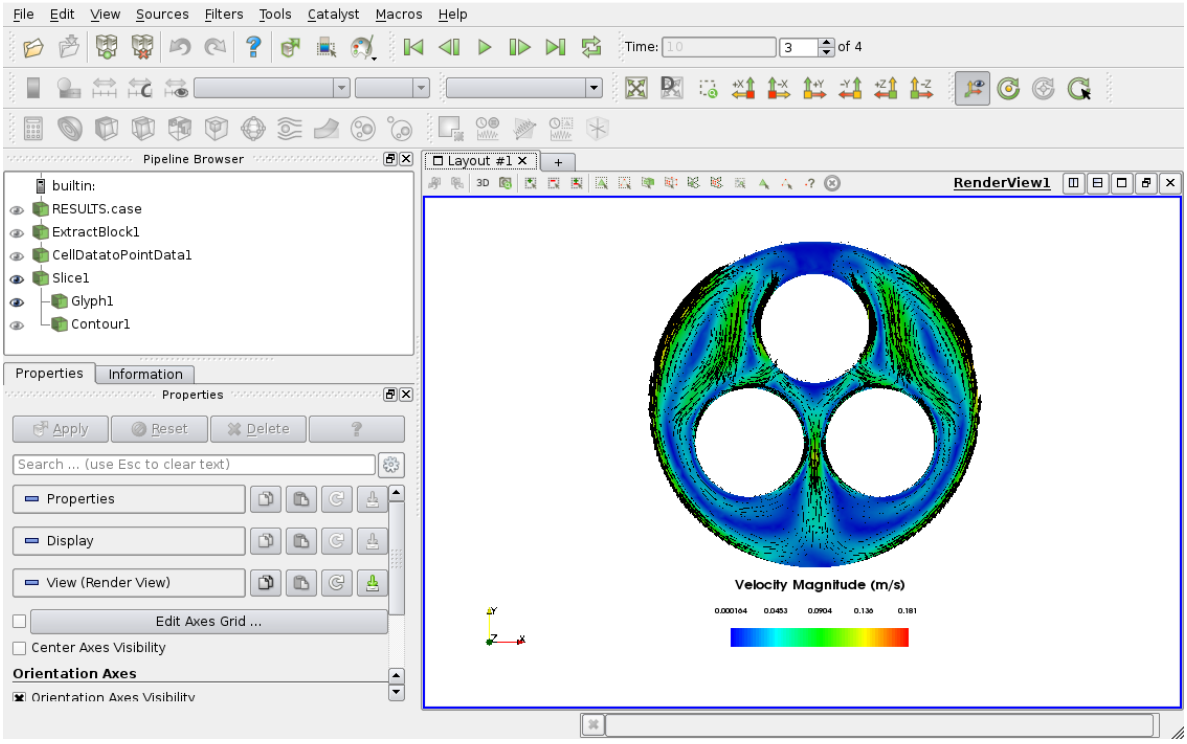

Figure III.29: Visualization of the 2D fluid velocity field

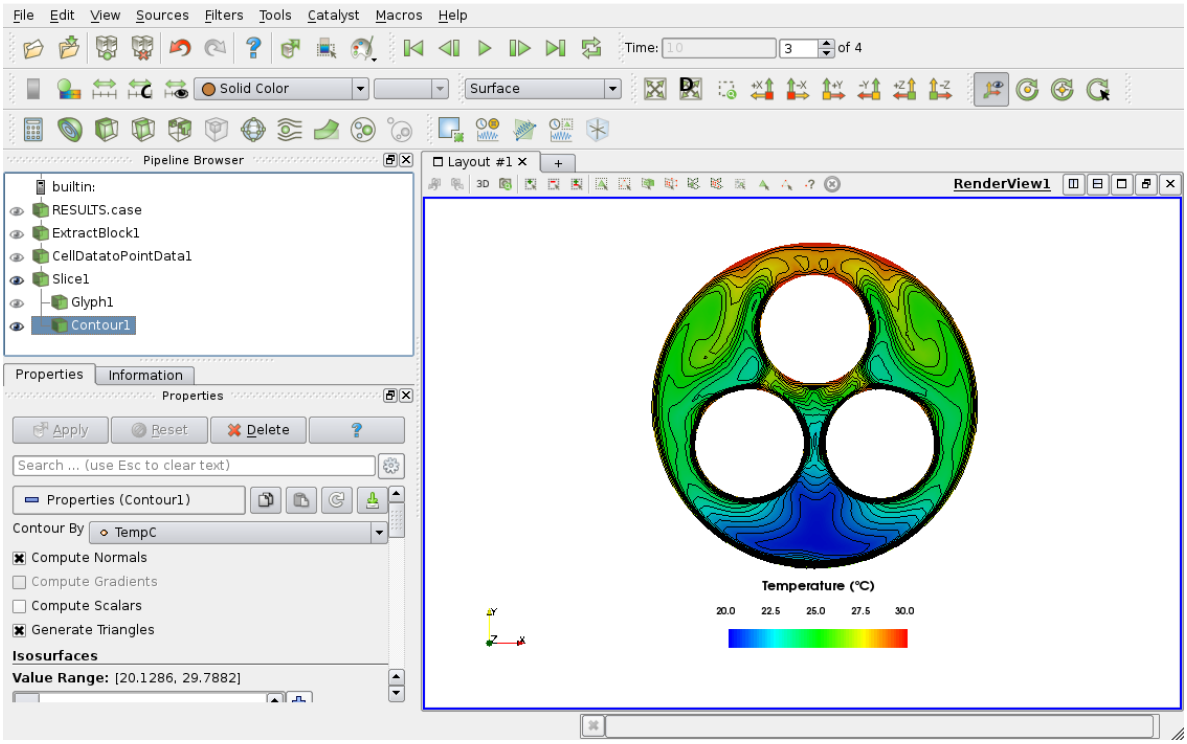

Figure III.30: Visualization of the 2D fluid temperature field

1.4 Preparing and launching *Code_Saturne*-SYRTHES coupled computation

The last modification to prepare the coupling computation are given below:
• **Step 1**: Activate conjugate heat transfer in SYRTHES GUI,
• **Step 2**: Activate conjugate heat transfer in *Code_Saturne* GUI,
• **Step 3**: Give identical number of iterations and reference time step for both codes,
• **Step 4**: Check the `coupling_parameters.py` python script and launch the calculation by executing the `runcase`.

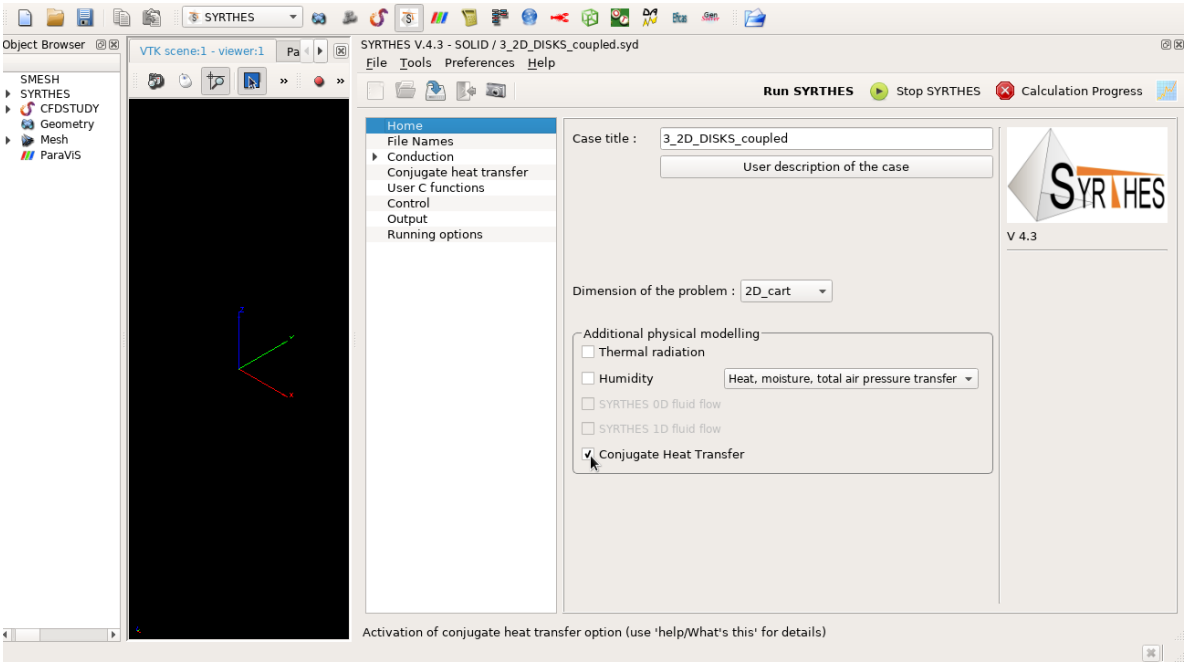

Figure III.31: Activate the conjugate heat transfer for the solid domain.

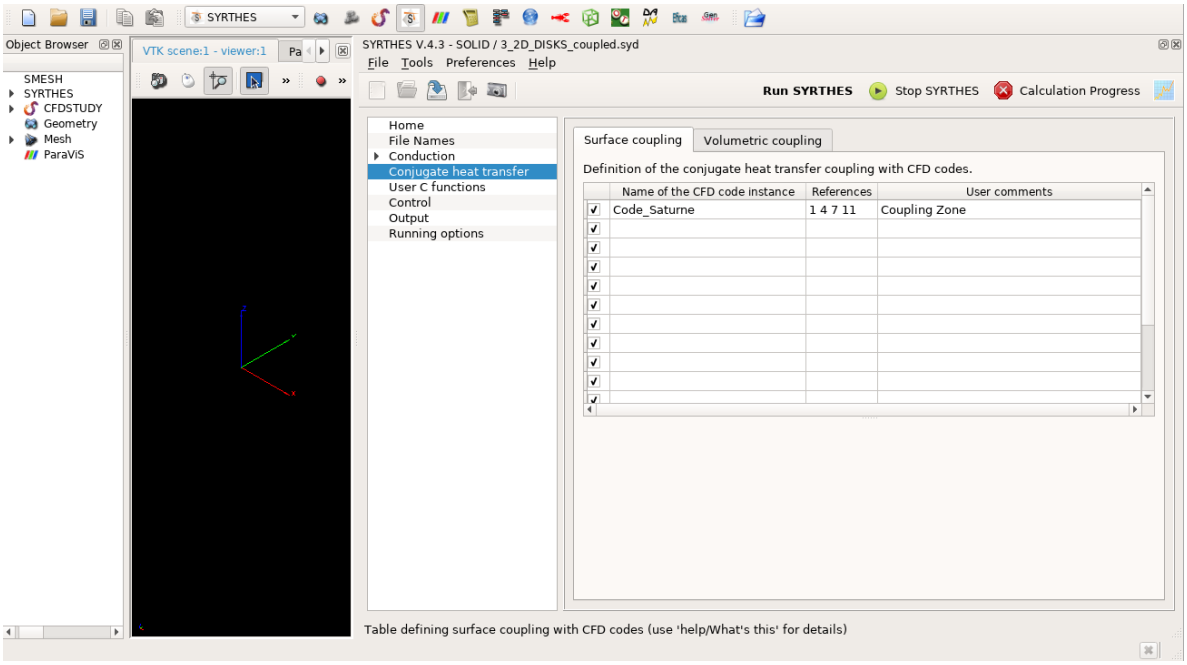

Figure III.32: Specify the references of the boundary zones for the coupling surfaces with *Code_Saturne*.

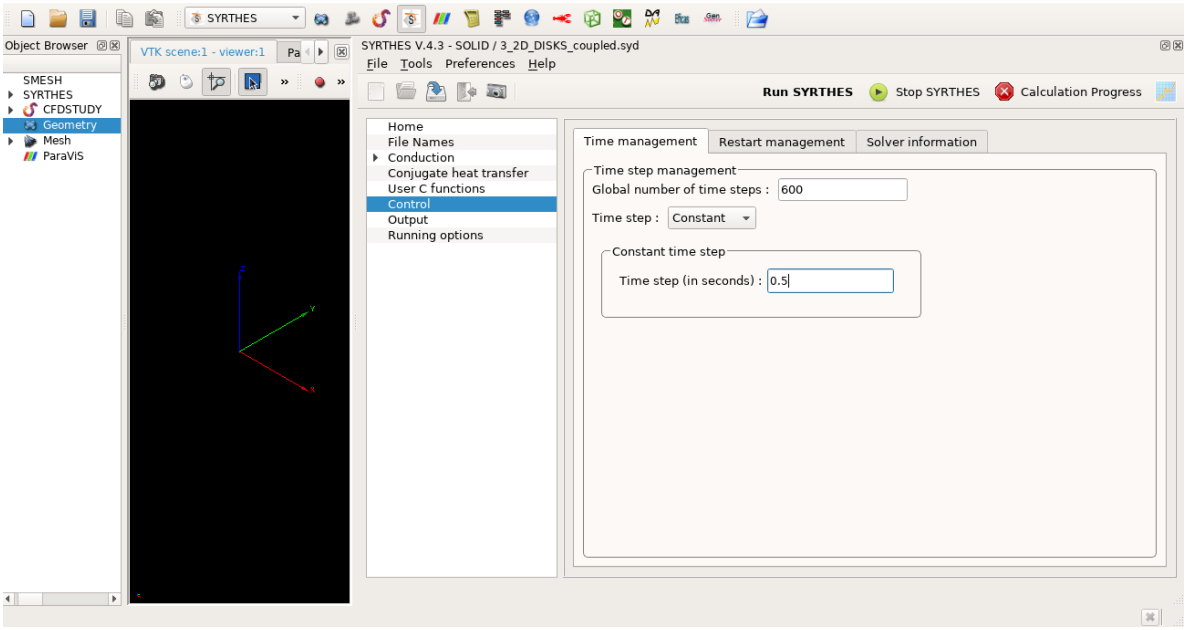

Figure III.33: Change the number of iterations and reference time step for the solid domain.

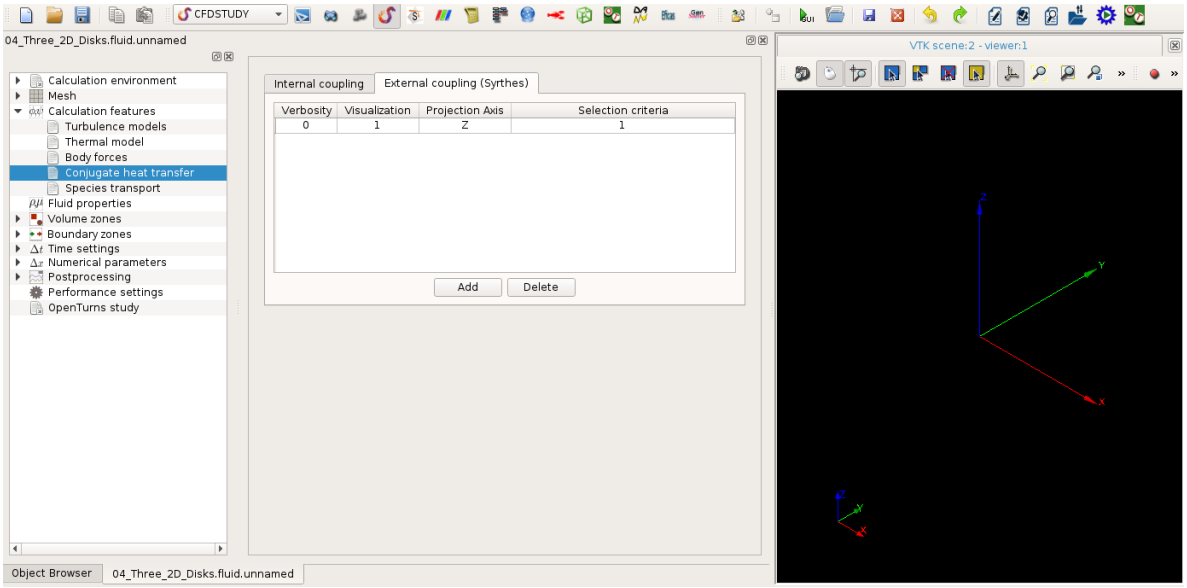

Figure III.34: Activate the conjugate heat transfer for the fluid domain.

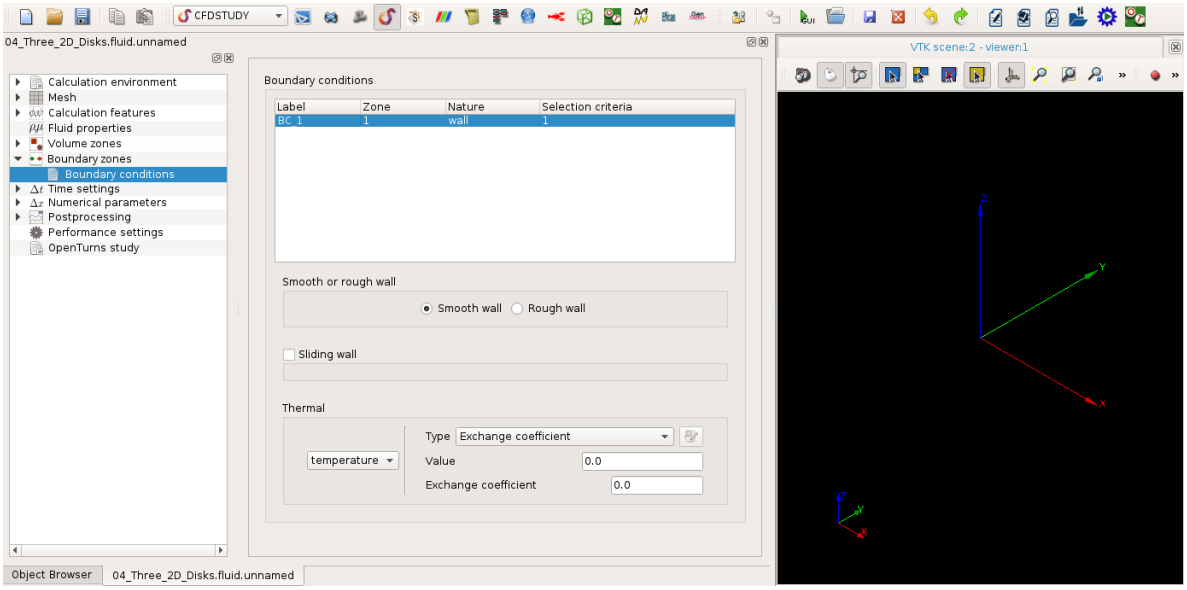

Figure III.35: Change the boundary conditions for the wall temperature.

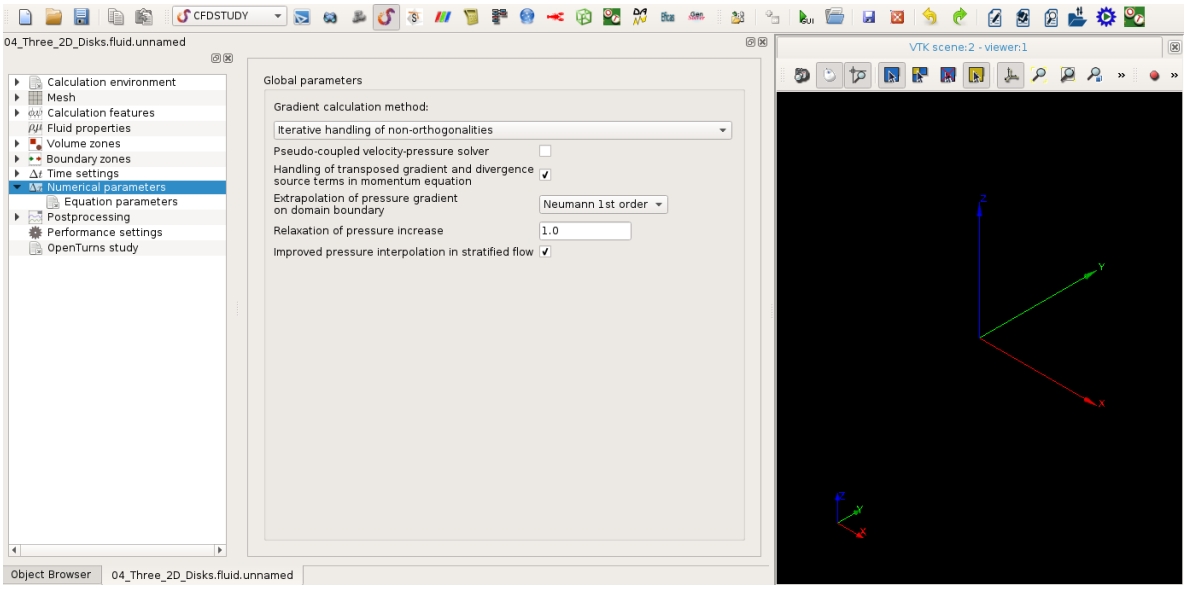

Figure III.36: Activate the **Improved pressure interpolation in stratified flow** algorithm.

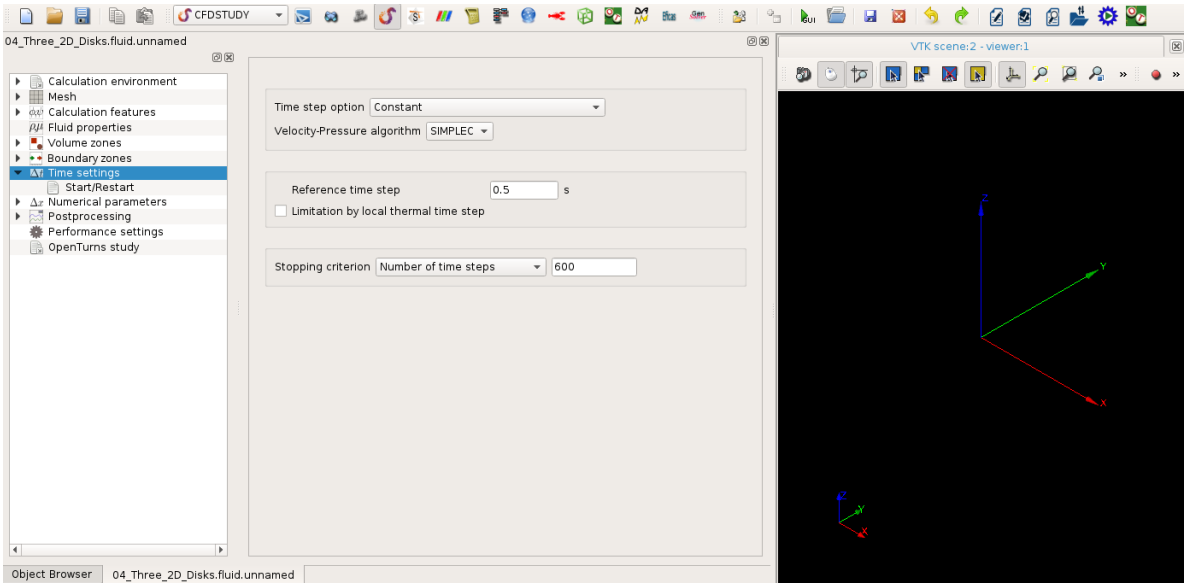

Figure III.37: Change the number of iterations and reference time step for the fluid computation.

• **Remark**: After having enabled conjugate heat transfer on both sides (in the fluid and in the solid data set), just increase the number of iterations (set it equal on both sides) and check the `coupling_parameters.py` script (set the number of procs for each side).

It is just needed to edit the `coupling_parameters.py` script and give the name of your SYRTHES script saved in the SYRTHES GUI as below:

```
 $ vim coupling_parameters.py
> domains = [
>
> 'solver':  'Code_Saturne',
> 'domain':  'FLUID',
> 'script':  'runcase',
> 'n_procs_weight':  None,
> 'n_procs_min':  4,
> 'n_procs_max':  4
>
> 'solver':  'SYRTHES',
> 'domain':  'SOLID',
> 'script':  '3_2D_DISKS_coupled.syd',
> 'n_procs_weight':  None,
> 'n_procs_min':  2,
> 'n_procs_max':  2,
> 'opt' :  '-v ens'
>
> ]
```

Finally, launch the `runcase` present in the study directory (named in our case 🗀 `3_2D_DISKS`) and run the coupling computation, as follows:

```
 $ runcase
```

• **Remarks**: in the `coupling_parameters.py`, the number of processors can be specified for each code (as this example with 4 processors for *Code_Saturne* and 2 processors for SYRTHES). It can be either both codes in parallel, one in parallel and the other one in sequential, or both in sequential.

One can specify the ouput results format for SYRTHES with an option (`opt`) which takes the value `-v ens` for a 3D fields output with a EnSight format or `-v med` for a 3D fields output with a SALOME format).